Defence Research and Development Canada
Recherche et développement pour la défense Canada

DEFENCE R&D DÉFENSE

Numerical Characterization of Mine Blast Loading

Final Report

L. Martin and R. Link
Martec Limited
Halifax, NS

Contract Scientific Authority: S.L. Hlady
DRDC Suffield

The scientific or technical validity of this Contract Report is entirely the responsibility of the contractor and the contents do not necessarily have the approval or endorsement of Defence R&D Canada.

Contract Report
DRDC Suffield CR 2006-016
August 2003

Canada

Numerical Characterization of Mine Blast Loading

Final Report

L. Martin and R. Link
Martec Limited

Martec Limited
Suite 400 – 1888 Brunswick Street
Halifax, NS B3J 3J8

Contract Number: W7702-01-R883

Contract Scientific Authority: S.L. Hlady

The scientific or technical validity of this Contract Report is entirely the responsibility of the contractor and the contents do not necessarily have the approval or endorsement of Defence R&D Canada.

Defence R&D Canada – Suffield

Contract Report

DRDC Suffield CR 2006-016

August 2003

© Martec Limited, 2003

Abstract

This report presents results from a numerical characterization of mine blast loading performed using experimental data obtained from DRDC Suffield small pendulum apparatus trials.

The validation of the IFSAS multiple material model was performed with data for two landmine trials representing the extremes in soil moisture content. The dependence of the applied impulse on the specific heat ratio and the soil stiffness was investigated.

Results from the landmine model are qualitatively and quantitatively consistent with experimental results. The trend of increasing impulse with increasing moisture content was reproduced. The specific heat ratio was found to have little effect on the calculated impulse. Also, the soil stiffness was decreased from its default value for sand in an attempt to determine the effect of other lower-stiffness soil components such as clay. Results indicate a significant dependence on soil stiffness, and demonstrate the need for more detailed soil information prior to simulating detonations in soils.

Executive summary

This report presents results from a numerical characterization of mine blast loading performed using experimental data obtained from DRDC Suffield small pendulum apparatus trials.

The validation of the IFSAS multiple material model was performed with data for two landmine trials representing the extremes in soil moisture content. The dependence of the applied impulse on the specific heat ratio and the soil stiffness was investigated.

Results from the landmine model are qualitatively and quantitatively consistent with experimental results. The trend of increasing impulse with increasing moisture content was reproduced. The specific heat ratio was found to have little effect on the calculated impulse. Also, the soil stiffness was decreased from its default value for sand in an attempt to determine the effect of other lower-stiffness soil components such as clay. Results indicate a significant dependence on soil stiffness, and demonstrate the need for more detailed soil information prior to simulating detonations in soils.

Martin, L. & Link, R. 2003. Numerical Characterization of Mine Blast Loading. DRDC Suffield CR 2006-016. DRDC Suffield.

106-04319/

MARTEC™

Smart Solutions for Engineering,
Science and Computing

Numerical Characterization of Mine Blast Loading

FINAL REPORT

Martec Technical Report # TR-03-18

August 11, 2003

Prepared by:

Laura Martin
Rick Link

W7702-01-R883
M 31203

Smart Solutions for Engineering,
Science & Computing

Martec Limited
1888 Brunswick Street, Suite 400
Halifax, Nova Scotia B3J 3J8 Canada

902.425.5101
902.421.1923
info@martec.com
martec.com

ABSTRACT

This report presents results from a numerical characterization of mine blast loading performed using experimental data obtained from DRDC Suffield small pendulum apparatus trials.

The validation of the IFSAS multiple material model was performed with data for two landmine trials representing the extremes in soil moisture content. The dependence of the applied impulse on the specific heat ratio and the soil stiffness was investigated.

Results from the landmine model are qualitatively and quantitatively consistent with experimental results. The trend of increasing impulse with increasing moisture content was reproduced. The specific heat ratio was found to have little effect on the calculated impulse. Also, the soil stiffness was decreased from its default value for sand in an attempt to determine the effect of other lower-stiffness soil components such as clay. Results indicate a significant dependence on soil stiffness, and demonstrate the need for more detailed soil information prior to simulating detonations in soils.

TABLE OF CONTENTS

1.0 Introduction 1

2.0 Problem Scope 1

3.0 Simulation Details 3
- 3.1 Balloon Analogue Calculation 4
- 3.2 Soil Composition Calculation 4
- 3.3 Initial Density Calculation 5

4.0 Validation Results 6

5.0 Conclusions 11

6.0 References 12

Appendices
- Appendix A - Ballistic Pendulum Mine Test Facility Report 13
- Appendix B - Multiple Material Model Theory 25
- Appendix C - Modifications to the LS-DYNA Pendulum Model 32
- Appendix D - FSI Tutorial 35
- Appendix E - IFSAS Tutorial 52
- Appendix F - Mesh Generator Manual 60

1.0 INTRODUCTION

This report presents results from a numerical characterization of mine blast loading performed using experimental data obtained from DRDC Suffield small pendulum apparatus trials.

The validation of the IFSAS multiple material model was performed with data for two landmine trials. In these trials, a 1 kg C4 landmine was placed with an overburden of 5 cm. The impulse applied to the pendulum at a standoff distance of 40 cm was measured. The two trials under consideration represent the lowest and the highest moisture contents for this test configuration.

Multiple simulations were run with this configuration to determine the effect of different model parameters on the resulting loads imparted to a pendulum target. In particular, the dependence of the applied impulse on the specific heat ratio and the soil stiffness was investigated.

Throughout this report, the term *soil* is used to denote a material composed of air, solid, and in the case of moist soil, water. The solid component was modelled as sand (quartz) for the initial calculations (density of 2650 kg/m^3 and a bulk modulus of 33.8 GPa). The properties for the solid component were modified in subsequent calculations in an attempt to account for other solid components present in the soil, such as clay.

2.0 PROBLEM SCOPE

The validation of IFSAS II's multiple material model was performed using experimental data obtained from trials using DRDC Suffield's small pendulum apparatus, pictured in Figure 1. The pendulum consists of a target plate attached to two rotating steel beams affixed to a stationary base. The pendulum target plate is suspended at some standoff distance over a buried landmine. The landmine is detonated, and the maximum angle of inclination obtained by the pendulum arm is measured, as shown in Figure 2.

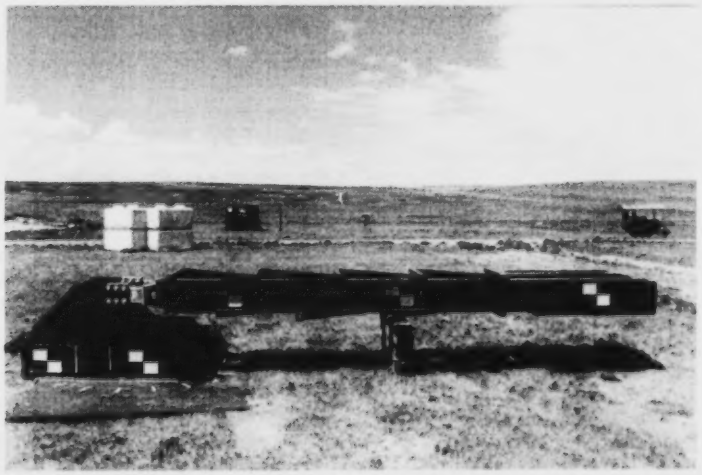


Figure 1: Small Pendulum Apparatus

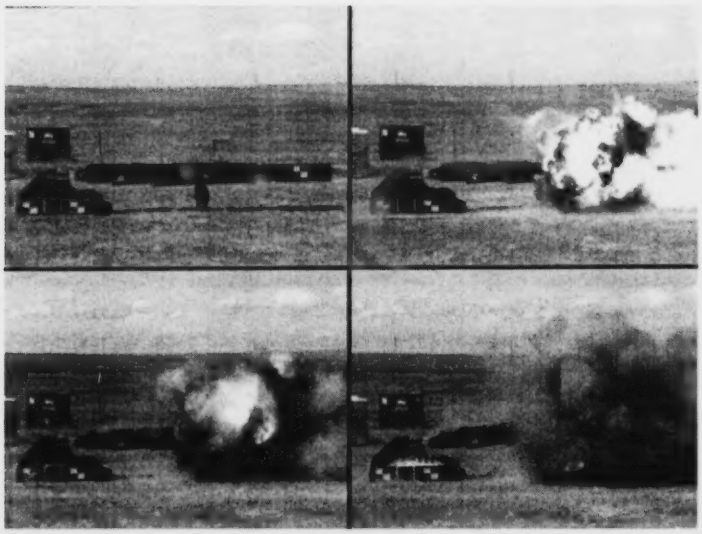


Figure 2: Rotation of Pendulum Arm with Landmine Detonation

The effective mechanical impulse applied to the target by the landmine can be expressed as a function of the maximum inclination of the pendulum. The following equation for the effective impulse was presented in Bergeron, Hlady, and Braid [1]. A derivation of this equation can be found in Whitehouse and Link [3], attached as Appendix A.

$$F_I = \sqrt{\frac{2mgrI_o \sin\theta}{R^2}} \tag{1}$$

where F_I = the net impulse applied to the pendulum
m = mass of the pendulum
g = gravitational constant
r = distance from the pivot point to the centre of mass
I_o = pendulum's mass moment of inertia about its pivot point
θ = the maximum angle of inclination
R = distance from the pivot point to the force application point

Previously, landmine simulations in IFSAS were performed using a dense multiphase model [4]. This model was limited in that only simulations involving dry soil could be performed. The multiple material model was created using a mixed cell approach whereby the pressure in a cell is calculated according to the cell mixed density and mass fractions of air, water and solid. Solids are not modelled explicitly as particles. Cells are treated as a continuum with all three phases having the same velocity and pressure. The gas phase is solved using the ideal gas equation of state (EOS), the water phase is solved using the Tait EOS, and the solid phase uses a linear-elastic EOS. Pressures less than 1 atm are solved using a modified form of the water and solid phase equations of state. The theory pertaining to this model can be found in Appendix B.

An LS-DYNA finite element model of the pendulum was created as part of a previous contract. There were some modifications to be made to this model which have been addressed as part of the current contract. The changes made to the pendulum model are discussed in Appendix C.

3.0 SIMULATION DETAILS

The code validation was performed with data for two landmine trials using the small pendulum apparatus. In these trials, a 1 kg C4 landmine was covered with an overburden of 5 cm. The impulse applied to the pendulum at a standoff distance of 40 cm was measured. The two trials under consideration represent the lowest and the highest moisture contents for this test configuration.

Figure 3 shows the setup for the CFD simulations. The charge is represented by a volume of high pressure gas of equal energy, and is placed at the appropriate depth. A solid plate is suspended above the ground, and has an area equivalent to the square experimental target plate although it is modelled as a disk due to the required axis of symmetry.

In the present work, the fluid solution is not coupled with a finite element analysis code such as LS-DYNA. Calculations of force applied to the target plate are performed using the IFSAS thrust calculator feature. Martec has developed the capability of performing FSI (fluid-structure interaction) problems; see Appendix D for a tutorial problem.

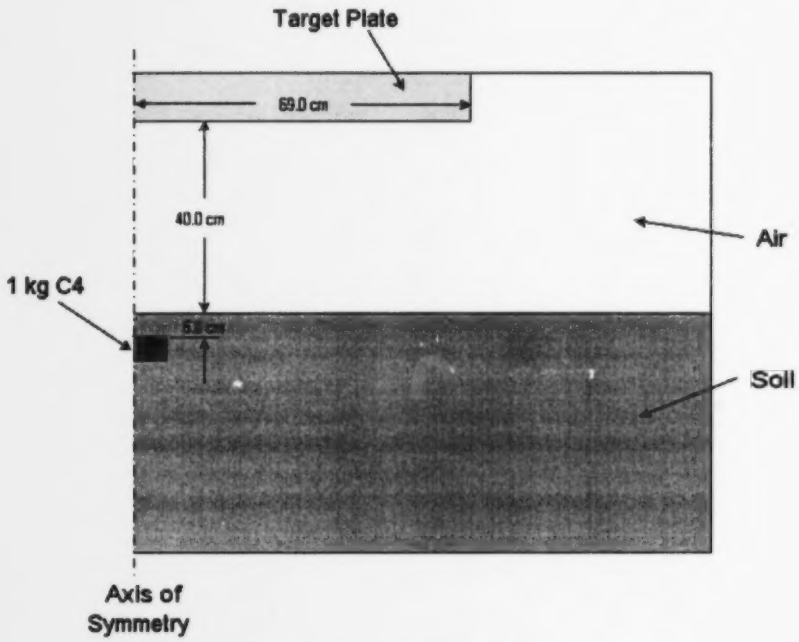


Figure 3: Model Setup

In the sections which follow, the calculation of the balloon analogue properties and the soil composition are discussed.

3.1 BALLOON ANALOGUE CALCULATION

In this model, the mine will be represented by a volume of high pressure gas of energy equal to the energy of the C4 charge. The charge dimensions are to match the original dimensions as close as is possible given the resolution restrictions. Based on the given mine dimensions, the actual charge volume is:

$$V_{actual} = \pi r^2 h = \pi(65.75\text{ mm})^2(46\text{ mm}) = 6.247399\times10^5\text{ mm}^3 \quad (2)$$

where V is the volume, r is the mine radius and h is the mine height. The simulation will have a 10 mm cell size. Therefore, the simulated mine radius and height are 60 mm and 50 mm, respectively, giving a simulated mine volume of:

$$V_{simulated} = \pi r^2 h = \pi(60\text{ mm})^2(50\text{ mm}) = 5.654867\times10^5\text{ mm}^3 \quad (3)$$

which represents a difference of approximately 9% with the actual volume.

The energy of a 1kg C4 charge is:

$$E = me_{TNT}R_{C4} = (1\text{ kg})(4.77\times10^6\text{ J/kg})(1.2) = 5.724\times10^6\text{ J} \quad (4)$$

where E is the total charge energy, m is the charge mass, e_{TNT} is the specific energy of TNT, and R_{C4} is the TNT blast impulse equivalency factor for C4. Using this energy value and the simulated volume, the balloon analogue pressure was determined:

$$P = \frac{E(\gamma-1)}{V_{simulated}} + P_o = \frac{(5.724\times10^6\text{ J})(1.4-1)}{5.65487\times10^{-4}\text{ m}^3} + 1\text{ atm} = 39959.5\text{ atm} \quad (5)$$

where P is the balloon analogue pressure, P_o is the ambient pressure, and γ is the simulated specific heat ratio for the balloon analogue gas. The multiple material model as implemented in IFSAS II is temperature independent, and therefore a balloon temperature is not required.

3.2 SOIL COMPOSITION CALCULATION

The soil properties of the trials used for validation purposes are shown in Table 1. The minimum and maximum moisture content results were obtained from Shot ID M00213A on August 1, 2000, and Shot ID M00130A on May 10, 2000, respectively.

Table 1: Soil Properties for Simulated Trials

	Dry Density (kg/m^3)	Wet Density (kg/m^3)	% Moisture	Impulse (Ns)
Min Moisture Content	1286	1363	6.0	3257
Max Moisture Content	1322	1702	28.7	5796

Using the wet and dry density values, the volume fractions of the three phases (air, water, solid), composing the soil can be determined. The assumed density values for the three phases are shown in Table 2.

Table 2: Phase Densities

Phase	Density (kg/m^3)
Gas (Air)	1.18
Water	1000
Solid (Sand)	2650

Consider the case where the moisture content is at a minimum. The volume fraction, ϕ, of solid in the dry soil is calculated according to Equation (B.2):

$$\rho_{mix} = \rho_{sand}\phi_{solid} + \rho_{air}\phi_{gas} + \rho_{water}\phi_{water}$$
$$1286\ \text{kg/m}^3 = (2650\ \text{kg/m}^3)\phi_{solid} + (0)\phi_{gas} + (1000\ \text{kg/m}^3)(0)$$
$$\phi_{solid} = 0.485$$

The air density is neglected in the above calculation due to its insignificant value in comparison to the mixture density. Also, the water volume fraction is zero since this is a dry density calculation.

It is assumed that the solid volume fraction remains constant for the dry and wet density calculations (i.e. water is simply displacing air surrounding a constant volume of solid). Therefore, the water volume fraction can be calculated by again using Equation (B.2), this time applying it to the wet density. Once again, the air density is neglected.

$$\rho_{mix} = \rho_{sand}\phi_{solid} + \rho_{air}\phi_{gas} + \rho_{water}\phi_{water}$$
$$1363\ \text{kg/m}^3 = (2650\ \text{kg/m}^3)(0.485) + (0)\phi_{gas} + (1000\ \text{kg/m}^3)\phi_{water}$$
$$\phi_{water} = 0.078$$

Since the total volume fraction must equal 1, the gas volume fraction can be determined using:

$$\phi_{gas} = 1 - \phi_{solid} - \phi_{water} = 1 - 0.485 - 0.078 = 0.437$$

This approach to determining the soil properties was repeated for the maximum moisture content case. The calculated soil compositions by phase are summarized in Table 3.

Table 3: Summary of Soil Composition by Phase Data

Trial	Volume Fractions		
	Gas	Water	Solid
Min Moisture Content	0.437	0.078	0.485
Max Moisture Content	0.121	0.380	0.499

3.3 Initial Density Calculation

The multiple material model in IFSAS 2002 requires the calculation of the gas initial density. For energy conservation the following equation must be valid:

$$\frac{E(\gamma-1)}{V} = P_o\left(1-\alpha\left(\frac{\rho_{exp}}{\rho_o}\right)^{\gamma}\right) + \alpha P_o \qquad (6)$$

where ρ_o is the initial density, α is the residual pressure factor equal to 0.01, and ρ_{exp} is the density of the explosive equal to:

$$\rho_{exp} = \frac{m}{V_{simulated}} \tag{7}$$

When Equation (6) is solved for ρ_o using a specific heat ratio of 1.4, the initial density for energy conservation is 0.906 kg/m^3.

4.0 VALIDATION RESULTS

IFSAS models for the two moisture content cases were created using the values calculated in the previous section. Details on the creation of the minimum moisture content model is outlined in Appendix E.

The force and impulse on the target plate for a specific heat ratio of 1.4 are shown in Figure 4 and Figure 5 for the minimum and maximum moisture content trials, respectively. The simulated data follows the correct trend; the impulse on the target is increased as the soil moisture content is increased. However, while the impulse in the minimum moisture content trial is within approximately 5% of the experimental value, the impulse was over-predicted in the maximum moisture case.

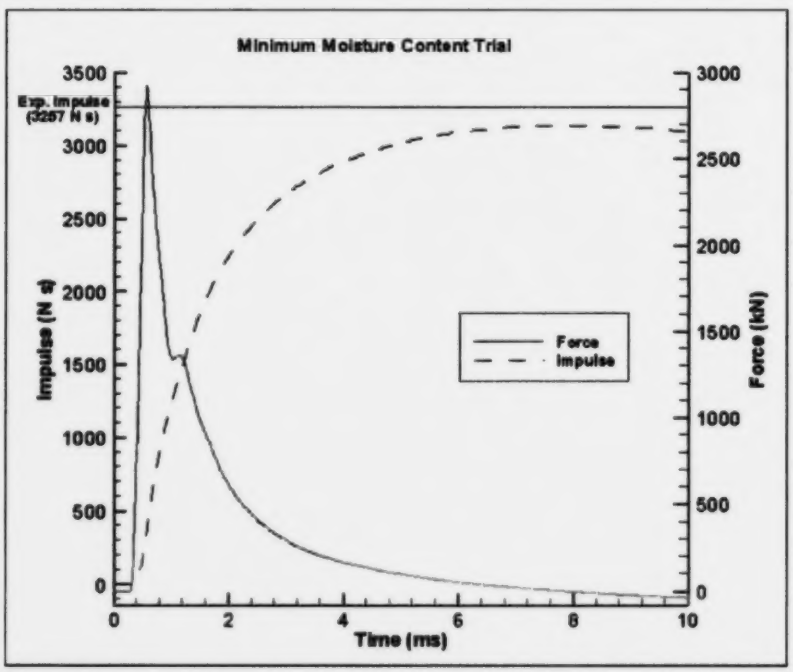


Figure 4: Impulse and Force on Target Plate for Minimum Moisture Content Trial

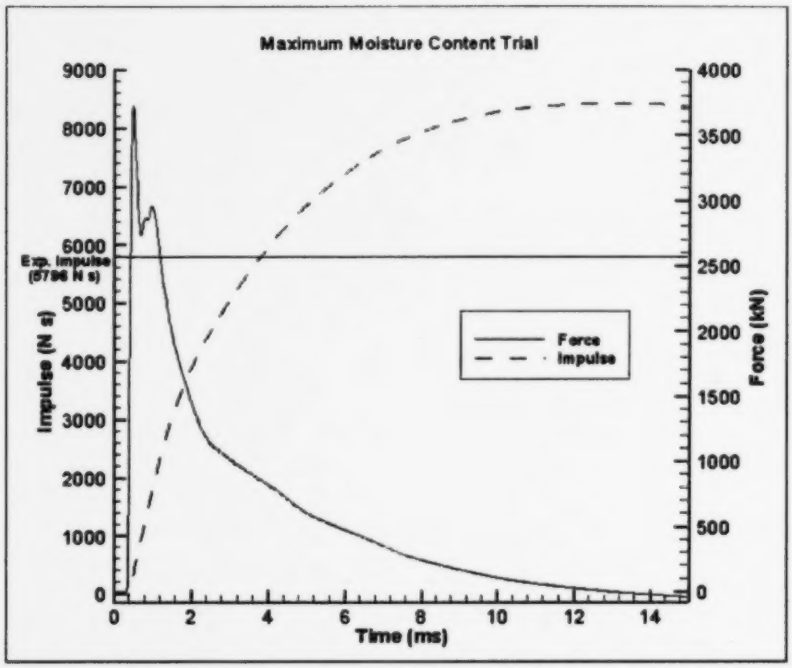


Figure 5: Impulse and Force Results for Maximum Moisture Content Trial

The specific heat ratio was modified in an attempt to achieve simulation results in closer agreement to the experimental results. Figure 6 and Figure 7 show the variance in impulse as the specific heat ratio is changed. As demonstrated in these plots, the specific heat ratio does not have a significant effect on the calculated impulse. Therefore, the over-prediction of the impulse appears to be a momentum transfer issue as opposed to a gas dynamics loading problem.

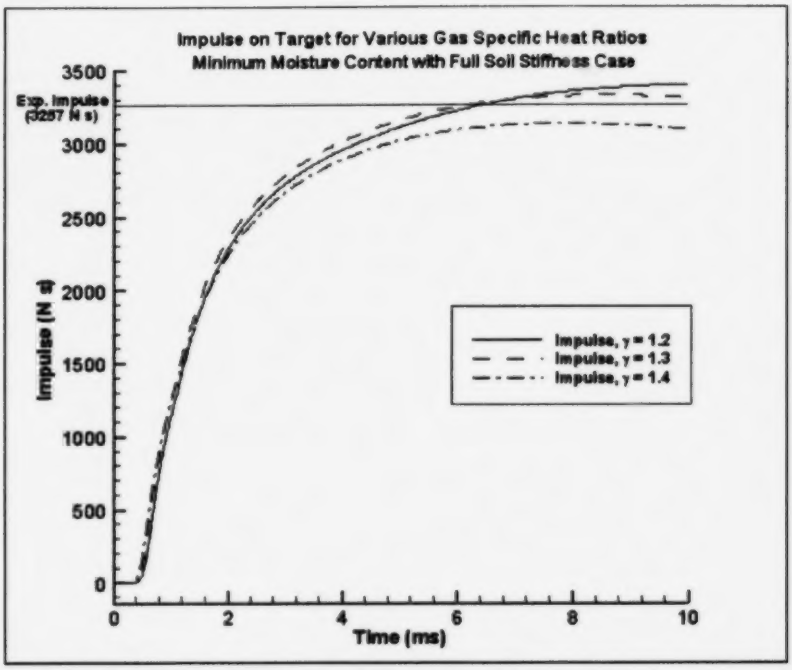

Impulse on Target for Various Gas Specific Heat Ratios
Minimum Moisture Content with Full Soil Stiffness Case
Exp. Impulse
(3257 N s)
Impulse (N s)
Time (ms)
Impulse, γ = 1.2
Impulse, γ = 1.3
Impulse, γ = 1.4

Figure 6: Effect of Specific Heat Ratio on Impulse for Minimum Moisture Content Trial

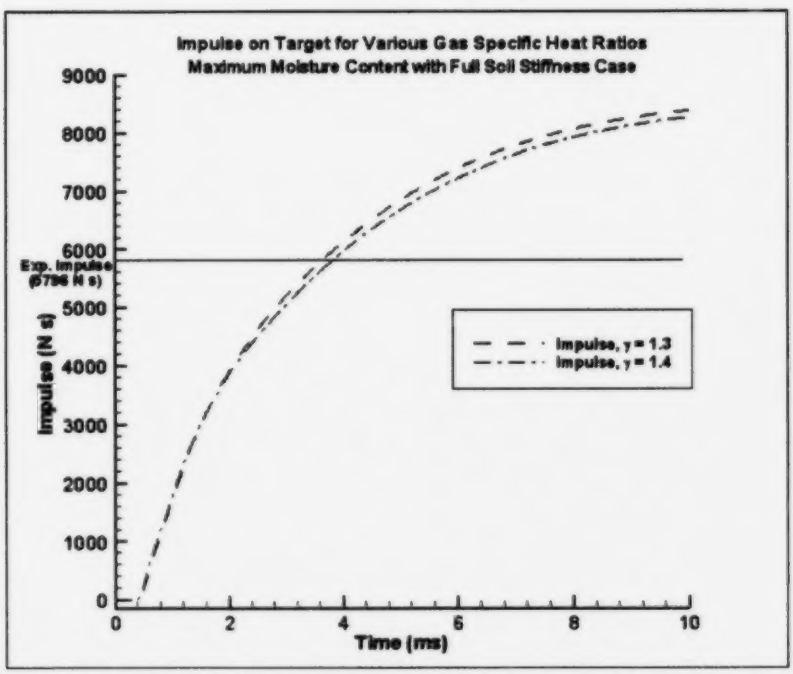

Impulse on Target for Various Gas Specific Heat Ratios
Maximum Moisture Content with Full Soil Stiffness Case
Exp. Impulse
(6796 N s)
Impulse (N s)
Time (ms)
Impulse, γ = 1.3
Impulse, γ = 1.4

Figure 7: Effect of Specific Heat Ratio on Impulse for Maximum Moisture Content Trial

The above results were generated using the default solid bulk modulus of approximately 33.8 GPa, the bulk modulus of sand. The experimental observations indicate that the soil composition is 40-60% sand and 10-20% clay. Clay is a softer material than sand and as a result a lower stiffness is expected. Therefore, a decrease in the overall stiffness of the soil was simulated in order to determine the effect on simulated impulse values.

Figure 8 and Figure 9 show the results obtained when the default solid bulk modulus is decreased by a factor of 1000 to determine the extent that the soil composition (in particular the clay content), may have on the simulated impulse. Comparison of these results with Figure 4 and Figure 5 demonstrate the significant effect of soil composition. The minimum moisture content case still shows a difference in impulse of close to 15%, however the impulse in the maximum moisture case is in better agreement with the experimental values.

The effect of soil stiffness is more pronounced in the soil containing the higher moisture content. The overall soil stiffness is higher in this case because the highly compressible air is displaced by the relatively incompressible water. In a low moisture content (high air content) soil, the solid stiffness plays a lesser role in the amount of impulse directed at the target since the air compression absorbs such a large amount of energy. This is evident in a comparison of Figure 4 and Figure 8. In high moisture content (low air content) soil mixtures, the stiffness of the solid component becomes more significant in determining the amount of impulse directed at the target, as evidenced by Figure 5 and Figure 9. In Figure 9, the more compressible, less stiff solid absorbs more energy hence transferring less impulse to the target than in Figure 5.

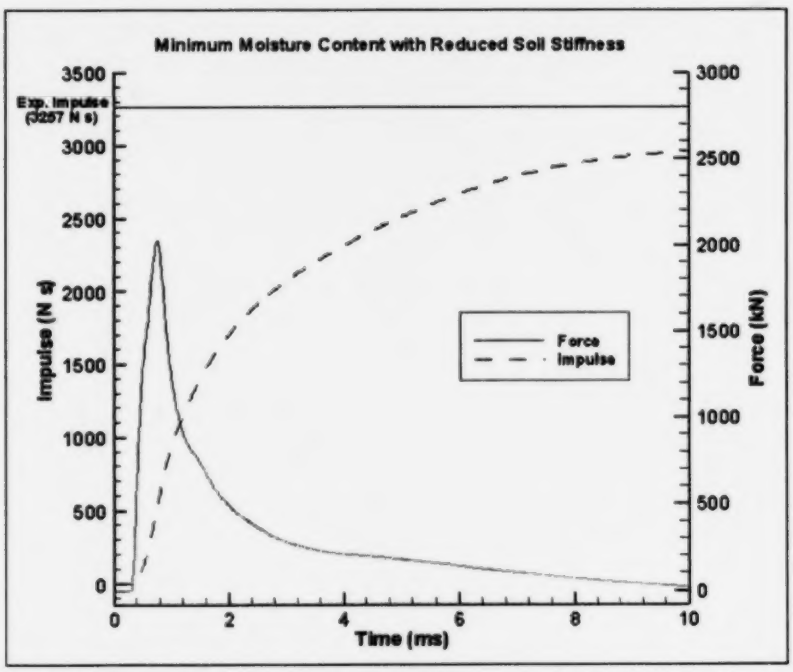


Figure 8: Force and Impulse Results for Minimum Moisture Content Trial Using Reduced Soil Stiffness

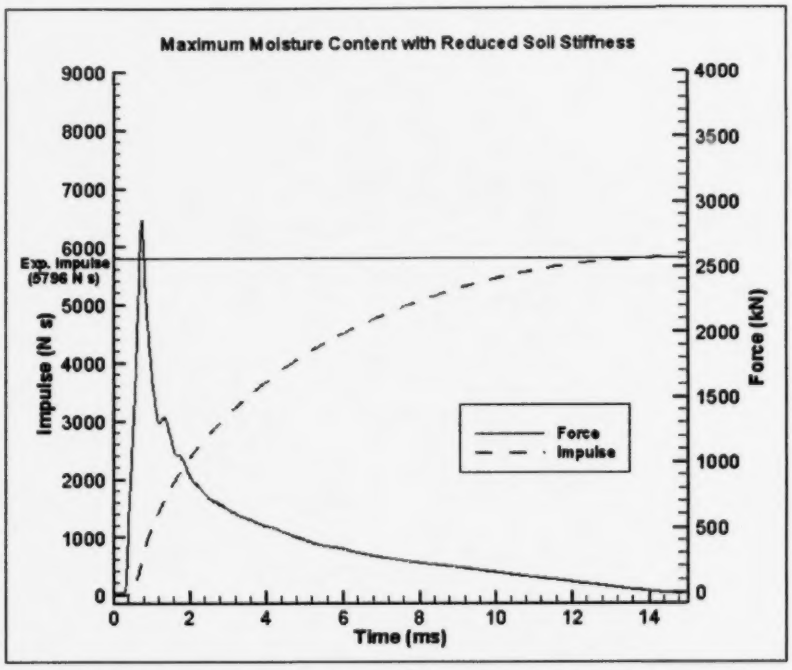


Figure 9: Force and Impulse Results for Maximum Moisture Content Case Using Reduced Soil Stiffness

All results continue to show the expected trend that the impulse applied to a target plate is enhanced as the soil moisture content is increased (Figure 10). However, these comparisons also demonstrate the need for more detailed soil information prior to attempting to simulate detonations in soils. In particular, data from a triaxial test, or other compression test, would allow the solid bulk modulus to be calculated. Such information is crucial in generating accurate results using CFD.

It should also be noted that no attempt was made in the present work to adjust any of the default parameters to achieve a result in closer agreement with experimental values. While the reduction in the soil stiffness did yield simulation results in very good agreement with experimental values, the change in the soil stiffness was made simply to gain an appreciation for the effect of soil stiffness on the total impulse applied to the target. Also, alterations were made to the FCT solver parameters to make the solution more diffuse and avoid instabilities which were causing difficulties running the simulation to completion.

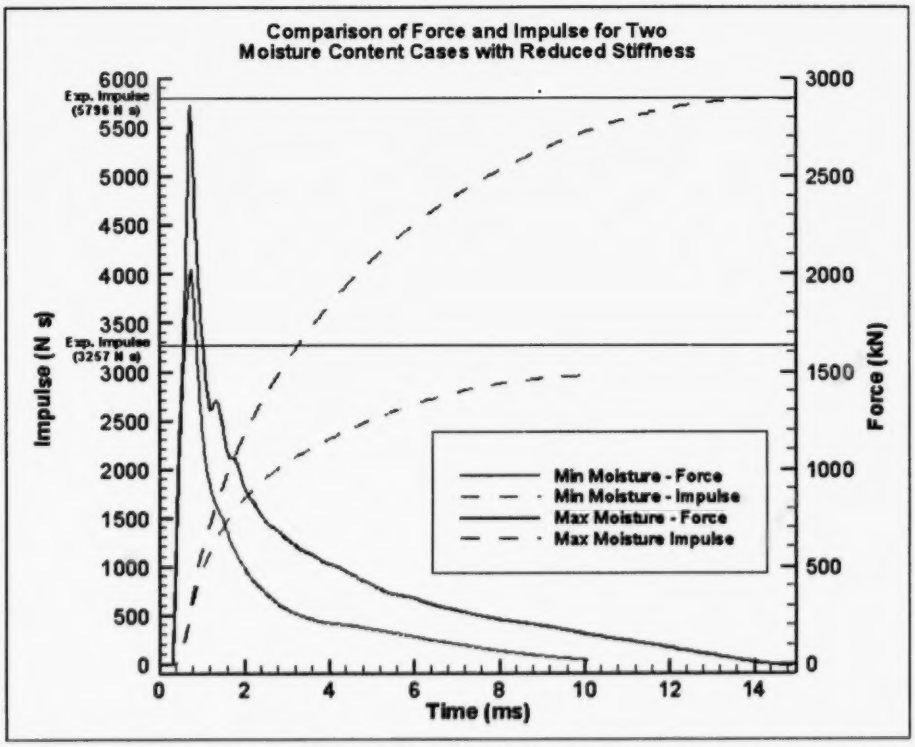


Figure 10: Comparison of Force and Impulse Results for Two Moisture Content Cases with Reduced Stiffness

5.0 CONCLUSIONS

The code validation was performed with data for two landmine trials using the small pendulum apparatus. In these trials, a 1 kg C4 landmine was placed with an overburden of 5 cm. The impulse applied to the pendulum at a standoff distance of 40 cm was measured. The two trials under consideration represent the lowest and the highest moisture contents for this test configuration.

Results from the landmine model are qualitatively and quantitatively consistent with experimental results. All results reproduce the correct trend of increased impulse for increased moisture contents. It was determined that when simulations were performed with the assumption that the soil bulk modulus was equivalent to that of sand, the impulse was over-predicted in the maximum moisture content case. When the soil bulk modulus was decreased to provide an indication of how other soil components, such as clay, would affect the results, a significant decrease in the applied impulse in the maximum moisture content case was observed. Therefore, information on the soil bulk modulus obtained from a compression test, such as a triaxial test, is required to properly simulate mine blast loading. It was also found that in comparison to the effect of soil stiffness on the impulse, altering the specific heat ratio had very little effect.

6.0 REFERENCES

[1] Bergeron, D.M., Hlady, S.L., Braid, M.P., "Pendulum Technique to Measure Land Mine Blast Loading", 17th Military Aspects of Blast Symposium, Las Vegas, Nevada, June 2002.

[2] Fairlie, G., Bergeron, D. "Numerical Simulation of Mine Blast Loading on Structures", 17th Military Aspects of Blast Symposium, Las Vegas, Nevada, June 2002.

[3] Whitehouse, D., Link, R.A., "Ballistic Pendulum Mine Test Facility - CFD and Rigid Body Motion Analysis", CDL Document, Final Report: February 1998.

[4] Hawken, D., Martin, L., Whitehouse, D., Wallace, P., Penrose, J., "Landmine Simulation Using Dense Multiphase Model", Technical Report #TR-03-13 submitted to DRDC Suffield, March 2003.

[5] Link, R.A., *Program freda - A Mesh Generation Program for Finite Element Meshes and CFD Grids*, Martec Ltd, June 2001.

[6] Combustion Dynamics Group, "Mesh Generation", Software Manual #SM-03-05, Martec Ltd., March 2003.

[7] Combustion Dynamics Group, "Fluid Structure Interaction Using IFSAS and LSDYNA", Software Manual #SM-03-06, Martec Ltd., March 2003.

[8] Combustion Dynamics Group, "Practical Modelling of High Explosives with CFD: Profile Mapping", Software Manual #SM-03-04, Martec Ltd., February 2003.

APPENDIX A BALLISTIC PENDULUM MINE TEST FACILITY REPORT

The following document is a report submitted to DRDC Suffield in 1998 outlining analysis of the pendulum test apparatus.

Ballistic Pendulum Mine Test Facility

CFD and Rigid Body Motion Analysis

W7702-5-R548 Phase III

by

David R. Whitehouse and Rick Link

Combustion Dynamics Limited

February 25, 1998

1. Introduction

This report describes results from the analysis of a pendulum test apparatus subjected to mine loads. A schematic of the pendulum is shown in Figure 1.1. The test configuration consists of a mine buried below the pendulum target plate. In this study, the system was analyzed using an uncoupled fluid-structure interaction model. Pendulum target plate loading was obtained using a two-dimensional multiphase CFD model assuming a rigid, stationary target surface. The loading was then used in a three-dimensional rigid body code assuming the body was free to rotate at the pendulum hinge. The calculations were performed using the IFSAS CFD module and a three-dimensional rigid body solver developed at CDL. The rigid body code allows the calculation of a general three-dimensional structure consisting of rigid bodies connected by springs.

Section 2 describes the results of a code validation performed on the rigid body code. Section 3 describes the pendulum apparatus and the pendulum analysis, including modeling considerations and results.

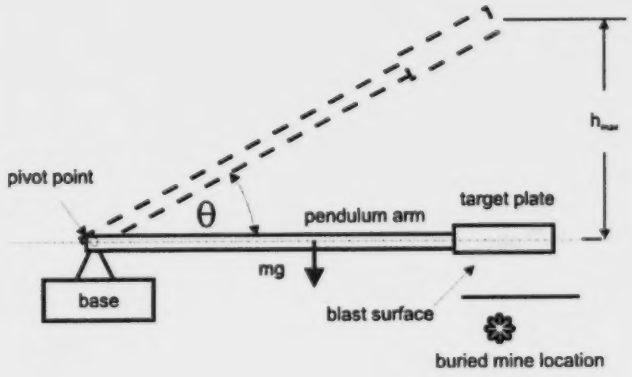


Figure 1.1 Pendulum schematic.

2. Rigid Body Code Validation

Before performing detailed calculations with the pendulum test apparatus, an analytic method was derived based on energy considerations that calculates the pendulum arm rotation, θ, as a function of the applied impulse and the pendulum properties. The details of this derivation are included in Appendix A. Table 2.1 shows results for a series of rigid body calculations and their corresponding analytic calculation. The quantity F_I is the applied impulse, I_o is the arm inertia about the pivot point, and M is the pendulum arm mass. Excellent agreement between the analytic and rigid body calculation was obtained in all cases.

Test	F_I (N-s)	I_o (kg-m^2)	M (kg)	$\theta_{analytic}$	$\theta_{rigid\ body}$
1	7410.0	33861	999	28.40	28.28
2	9781.2	33861	999	56.01	55.67
3	7408.8	132950	999	14.02	13.95
4	14817.6	132950	999	75.77	75.63

Table 2.1 Comparison of analytic and rigid body calculations.

3. Pendulum Analysis

3.1 Pendulum Configuration

The ballistic pendulum apparatus is shown in Figure 3.1. The apparatus consists of:

1. A steel plate, laid flat on the ground, on which the pendulum apparatus is free to slide.
2. The pendulum base. A water filled steel reservoir with a pivot point for the pendulum arm.
3. The pendulum arm.
4. The target plate attached to the end of the pendulum arm

The mass and inertia of the pendulum apparatus are given in Table 3.1:

Part	MASS (kg)	INERTIA (kg-m^2)
PENDULUM BASE	2586	5616
PENDULUM ARM	878	13243

Table 3.1 Physical properties of the pendulum test apparatus.

The values for the pendulum arm were assumed to include the target plate. The moment of inertia of each body was assumed to be about the axis passing through the pivot point.

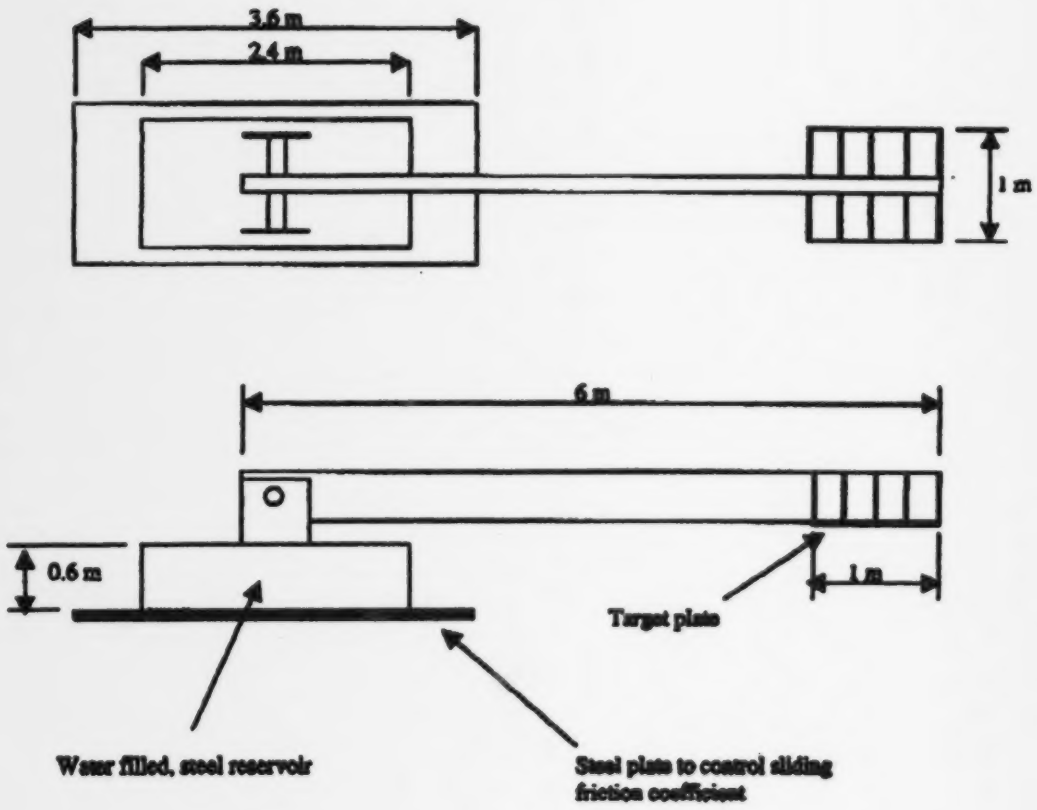


Figure 3.1 Rough sketch of the ballistic pendulum test apparatus.

3.2 Buried Mine Model

Figure 3.2 shows the CFD model setup. A 135 mm diameter, 45 mm thick cylindrical disk of 1 kg C4 charge was buried in the ground. A rigid cylindrical disk 0.564 m in diameter was fixed 0.6 m above the ground surface, and is meant to represent the pendulum target plate. The disk had the same surface area as the pendulum arm's target plate. The ground was composed of dry sand with an average particle diameter of 420 Om and a material density of 2670 kg/m^3. The packed sand had an initial volume

fraction of 0.614. The STATE and CFD model parameters for this calculation are provided in the report for Phase II of SSC Contract W7702-5-R548.

Eleven monitor points spaced 0.05 m apart were positioned on the bottom surface of the cylinder at distances from 0.05 m to 0.55 m from the axis of symmetry. After the calculation, the monitor point pressure data were saved to data files and processed to generate transient loading data for the rigid body calculation.

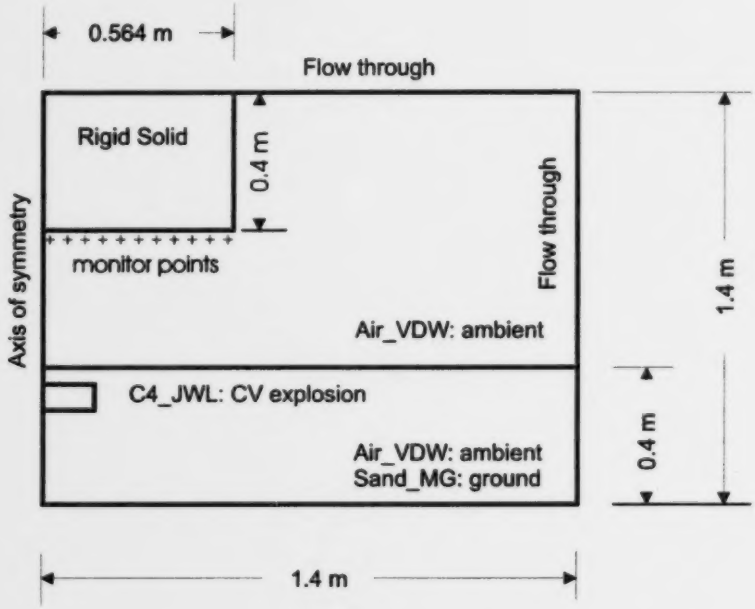


Figure 3.2 CFD setup of buried mine.

3.3 Pendulum Model

The rigid body pendulum model consists of two rigid bodies connected at a pivot point. The dimensions and physical properties of each body were selected to match the inertia and mass of the experimental apparatus as closely as possible.

Figure 3.3 shows the full three-dimensional pendulum model. During the initial calculation the pendulum base was prevented from moving. The pivot point was simulated using two springs with stiffness of 1.0×10^{14} N/m; one spring restricts relative movement between the bodies in the x_1 direction and the other restricts movement in the x_3 direction (see Figure 3.3). A third spring in the x_2 direction was unnecessary as the problem was symmetric about the x_1-x_3 plane. The net effect of these constraints is that when sufficient force is applied to the pendulum arm it will move about the pivot point defined by the springs.

The body surface is plotted using triangular elements. These elements are normally used for contact simulation which is not considered here.

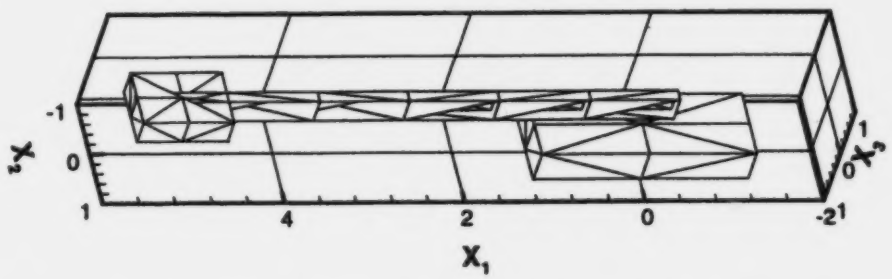


Figure 3.3 Three-dimensional ballistic pendulum rigid body model.

3.4 Results

3.4.1 Explosive Time History

The pendulum mine loading was applied as a concentrated load to the target plate. This transient load was calculated by integrating the static pressure data from the monitor points at each CFD time step over the surface area of the target plate. The transient force as a function of time is shown in Figure 3.4.

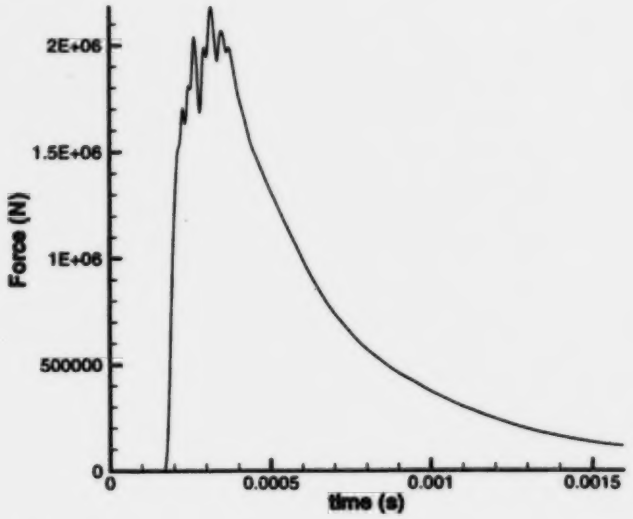


Figure 3.4 Force-time data calculated from CFD pressure data.

3.4.2 Pendulum Motion

The force data obtained from the CFD calculation was used in the rigid body code and integrated to obtain the impulse data needed for an analytic calculation. Table 3.3 compares results obtained from rigid body calculations and an analytic solution using Equation A.9 from Appendix A. The quantity θ is the pendulum arm rotation and h_{max} is the maximum height the far-end of the target plate reaches during the calculation as shown in Figure 1.1. It is seen that there is excellent agreement between both methods.

	Analytic Solution	Rigid Body Code
θ (degrees)	2.056	2.049
h_{max} (m)	0.2045	0.2037

Table 3.3 Results from rigid body code and analytic solution.

3.4.3 Base Forces

The amount of force transferred from the target plate to the pivot point during the impulsive loading can be calculated using Equation B.8. If the maximum force applied to the target plate by the forcing function (2.17×10^6 N) is selected, the resultant vertical load on the pivot point is 393,054 N acting upward. The rigid body code calculates a value of 390,716 N for this force.

4. Conclusions

A computer model was developed to obtain the response of a pendulum arm subjected to an impulsive mine load. The model utilized a one-way coupled fluid-structure interaction approach. The pendulum loading was computed using an axisymmetric CFD multiphase model. The pendulum movement was calculated using a three-dimensional rigid body model.

The rigid body model was validated by using simple analytic models derived from energy considerations. Close agreement was achieved between analytic models for the pendulum arm rotation and the vertical loading at the pivot point and the rigid body code results.

A methodology for the rigid body analysis of an experimental pendulum apparatus was described in this report. Pressure loads on pendulum target plate can be calculated using the IFSAS CFD module. The resulting transient force on the pendulum can be calculated and used with analytic methods presented in this report to calculate pendulum arm movement and loads on the pendulum base. This methodology can be used to assist in designing the experimental pendulum apparatus. It is also suitable for performing a parametric study of the ballistic pendulum problem.

5. References

Penrose, John, "Modelling of In-Soil Explosions and Interactions with Structures", Report for Phase II of SSC Contract W7702-5-R548, November 28, 1997.

Appendix A Analytic Pendulum Model

An analytic method for calculating the inclination of an ideal rigid pendulum starting from a horizontal position and subjected to a transient vertical load was derived. The analysis uses conservation of energy and assumes that all kinetic energy input has occurred before the pendulum starts moving. Friction forces are neglected.

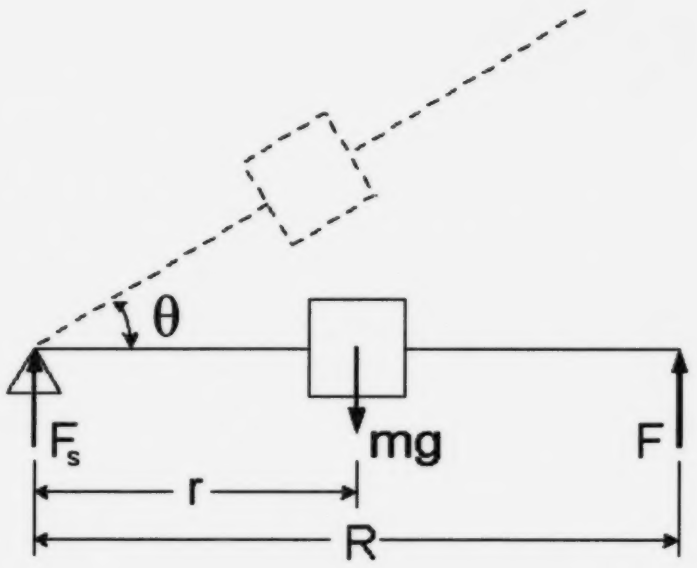


Figure A.1 Idealized pendulum model.

Applying conservation of potential and kinetic energy between the initial rest position and the final inclined position:

$$KE_1 + PE_1 = KE_2 + PE_2 \qquad \textbf{[A.1]}$$

Since the initial potential energy is zero and the final kinetic energy is zero the equation can be rewritten:

$$KE_1 = PE_2 \qquad \textbf{[A.2]}$$

The initial rotational kinetic energy, KE_1 can be written:

$$KE_1 = \frac{1}{2} I_o \omega_o^2 \qquad \text{[A.3]}$$

If all of the initial kinetic energy is assumed to be applied before the pendulum begins to move, the initial angular velocity, ω_o, can be calculated from the impulse relationship:

$$I_o \omega_o = \int \bar{F} \times \bar{R} dt \qquad \text{[A.4]}$$

$$I_o \omega_o = R \int F dt \qquad \text{[A.5]}$$

$$\omega_o = \frac{R}{I_o} \int F dt = \frac{R}{I_o} F_I \qquad \text{[A.6]}$$

Using this expression for the angular velocity the initial kinetic energy can be written:

$$KE_1 = \frac{1}{2} I_o \left(\frac{R}{I_o} F_I \right)^2 \qquad \text{[A.7]}$$

The final potential energy for the inclined pendulum can be written:

$$PE_2 = mgr \sin\theta \qquad \text{[A.8]}$$

Equating the expressions for the initial kinetic and final potential energy and solving for $\sin\theta$ gives the following:

$$\sin\theta = \frac{1}{2mg} \frac{R^2}{r} \frac{F_I^2}{I_o} \qquad \text{[A.9]}$$

where:

θ	angle of inclination.
F_I	net impulse applied to the pendulum.
m	mass of the pendulum
g	gravitational constant
I_o	pendulum's mass moment of inertia about its pivot point.
r	distance from the pivot point to the centre of mass.
R	distance from the pivot point to the force application point.

Appendix B Dynamic Reaction Force Analysis

The vertical reaction force at the pivot point during the impulsive loading can be evaluated using a dynamic analysis. Using the dimensions in Figure A.1 for the pendulum in its initial position we first sum moments about the pivot point to solve for the angular acceleration,α:

$$\sum M_o = I_o \alpha \qquad \textbf{[B.1]}$$

$$FR = I_o \alpha \qquad \textbf{[B.2]}$$

$$\alpha = \frac{FR}{I_o} \qquad \textbf{[B.3]}$$

Next, sum the forces in the vertical direction and solving for the vertical reaction force at the pivot point, F_s:

$$\sum F_y = ma \qquad \textbf{[B.4]}$$

$$F_s + F - mg = ma \qquad \textbf{[B.5]}$$

$$F_s = mr\alpha - F + mg \qquad \textbf{[B.6]}$$

Substituting for α in Equation B.6 with the expression from Equation B.3 we obtain the following:

$$F_s = mr\frac{FR}{I_o} - F + mg \qquad \textbf{[B.7]}$$

$$F_s = F(mr\frac{R}{I_o} - 1) + mg \qquad \textbf{[B.8]}$$

where:

F	force applied to the target plate
F_s	vertical reaction force at the pivot point
m	mass of the pendulum
g	gravitational constant
I_o	pendulum's mass moment of inertia about its pivot point.
r	distance from the pivot point to the centre of mass.
R	distance from the pivot point to the force application point.

APPENDIX B MULTIPLE MATERIAL MODEL THEORY

It is assumed in the multiple material model that all material components are subjected to the same pressure and the velocity.

Fluxes

$$F_{n_i} = V_{ni}\begin{Bmatrix} \rho_{mix}T_i \\ \rho_{mix}T_i u \\ \rho_{mix}T_i v \\ \rho_{mix}T_i w \end{Bmatrix} + P_{mix}\begin{Bmatrix} 0 \\ n_x \\ n_y \\ n_z \end{Bmatrix} \tag{B.1}$$

where V_n = velocity normal to a given cell face
u,v,w = material velocities
T = mass fraction
ρ = density
i = subscript to denote phase
n_x, n_y, n_z = face normals

and:

$$\rho_{mix} = \sum_{i=1}^{\#\text{ of phases}} \rho_i \phi_i \tag{B.2}$$

P_{mix} is the mixed pressure, to be determined by iteration, and ϕ is the volume fraction.

B.1 SOLUTION STRATEGY

All conservative variables are advected for each phase. Variables T_i and ρ_{mix} are then used to determine the mixed pressure.

Procedure

1) Solve Equation (B.2):

$$\rho_{mix} = \sum_{i=1}^{\#\text{ of phases}} \rho_i \phi_i$$

2) Calculate the mass fractions of the phases:

$$T_i = \frac{\rho_{mix}T_i}{\rho_{mix}} \tag{B.3}$$

3) Calculate the material velocities for each phase:

$$u = \frac{\rho_{mix} T_i u}{\rho_{mix} T_i} \tag{B.4a}$$

$$v = \frac{\rho_{mix} T_i v}{\rho_{mix} T_i} \tag{B.4b}$$

$$w = \frac{\rho_{mix} T_i w}{\rho_{mix} T_i} \tag{B.4c}$$

4) Compute P_{mix}, ρ_i from iteration using ρ_{mix} and T_i.

B.2 SOUND SPEED

For energy independent equations of state, the mixed sound speed, a_{mix}, is given by:

$$a_{mix} = \left(\frac{\partial P}{\partial \rho}\right)^{1/2}_{mix} \tag{B.5}$$

The $\frac{\partial P}{\partial \rho_{mix}}$ term will now be defined. From the chain rule we have:

$$\frac{\partial \rho_{mix}}{\partial P} = \sum \frac{\partial \rho_{mix}}{\partial \rho_i} \frac{\delta \rho_i}{\partial P} \tag{B.6}$$

Also, from Equation (B.2):

$$\rho_{mix} = \sum \phi_i \rho_i$$

and,

$$\frac{\partial \rho_{mix}}{\partial \rho_i} = \phi_i \tag{B.7}$$

Combining Equations (B.6) and (B.7) gives:

$$\frac{\partial \rho_{mix}}{\partial P} = \sum \phi_i \frac{\partial \rho_i}{\partial P} \tag{B.8}$$

Finally, taking the inverse of each side of Equation (B.8) yields:

$$\frac{\partial P}{\partial \rho_{mix}} = \frac{1}{\sum \frac{\phi_i}{\frac{\partial P}{\partial \rho_i}}} \tag{B.9}$$

B.3 SOLUTION PROCEDURE

The multiphase module consists of two parts:

1) Root bracketing - brackets pressure, P, by upper and lower pressures P_U and P_L.
2) Multiphase solver - solves for P_{mix}.

The bracketed pressure is solved for by using the method of false position, illustrated in Figure 11. P_r is equal to:

$$P_r = P_U - \frac{\rho_{mixU}(P_L - P_U)}{\rho_{mixL} - \rho_{mixU}} \tag{B.10}$$

Also defined are:

$$f_L = \rho_{mixL} - \rho_{mix} \tag{B.11a}$$
$$f_r = \rho_{mixr} - \rho_{mix} \tag{B.11b}$$

The brackets are set according to the condition:

$$\text{If } f_L \times f_r < 0 \qquad P_L = P_r \tag{B.12a}$$
$$\text{If } f_L \times f_r > 0 \qquad P_U = P_r \tag{B.12b}$$

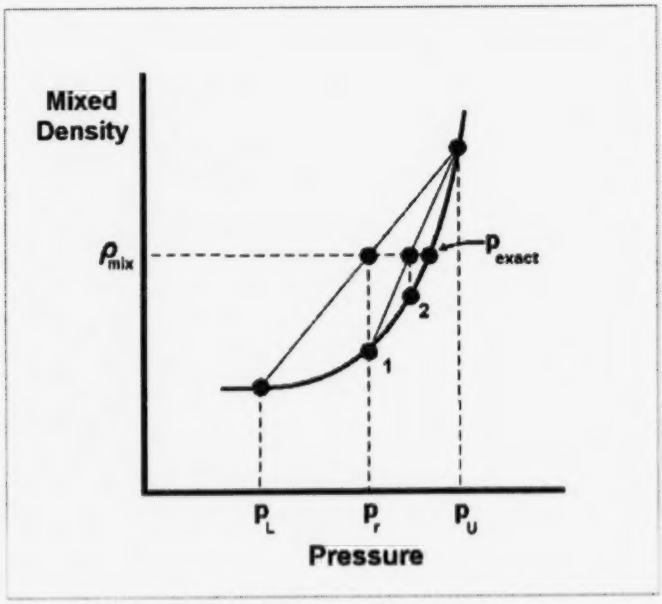


Figure 11: Method of False Position

B.4 ROOT BRACKETING PROCEDURE

1) Compute ρ_{iL} , ρ_{iV} from the equations of state.

2) Compute ϕ_{iL}, ϕ_{iU}:

$$\phi_{L,U} = \left(\frac{T_i \rho_{mix}}{\rho_i} \right)_{L,U} \tag{B.13}$$

3) Compute ρ_{mixL} , ρ_{mixV}:

$$\rho_{mixL,U} = \left(\sum_{i=1}^{\#\text{ of phases}} \rho_i \phi_i \right)_{L,U} \tag{B.14}$$

4) Check for bracket:

If $\rho_{mixL} \leq \rho_{mix}$ and $\rho_{mixU} \geq \rho_{mix}$, solution bracketed.

If not bracketed,

- If $\rho_{mix} \geq \rho_{mixU}$, shift P_L, P_U right
- If $\rho_{mix} < \rho_{mixL}$, shift P_L, P_U left
- Return to Step (1)

This procedure is illustrated in Figure 12. Once the root has been bracketed, the multiphase solver procedure begins.

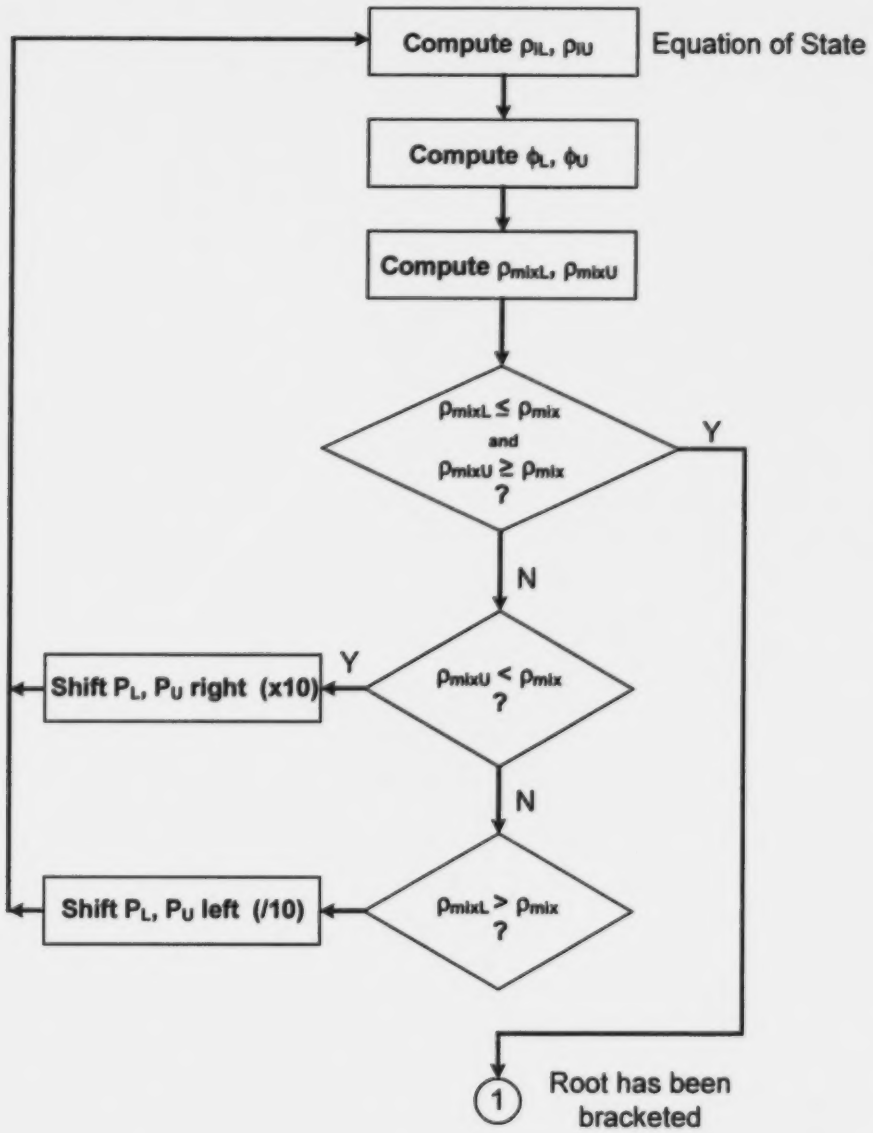


Figure 12: Root Bracketing Procedure

B.5 MULTIPHASE SOLVER PROCEDURE

1) Compute f_L, f_U according to Equations (B.11a) and (B.11b):

$$f_L = \rho_{mixL} - \rho_{mix}$$
$$f_r = \rho_{mixr} - \rho_{mix}$$

2) Compute P_r using Equation (B.10):

$$P_r = P_U - \frac{\rho_{mixU}(P_L - P_U)}{\rho_{mixL} - \rho_{mixU}}$$

3) Compute ρ_{mixr}:

$$\rho_{mixr} = \sum_{i=0}^{\#\text{ of phases}} \rho_{ri}(P_r)\phi_{ri} \tag{B.15}$$

where ϕ_{ri} is equal to:

$$\phi_{ri} = \frac{T_i \rho_{mix}}{\rho_i} \tag{B.16}$$

4) Set the brackets according to the condition in Equations (B.12a) and (B.12b):

If $f_L \times f_r < 0$ $\quad P_L = P_r$
If $f_L \times f_r > 0$ $\quad P_U = P_r$

5) Check the tolerance:

If $\frac{P^{i+1} - P^i}{P^{i+1}} <$ tolerance, solution is converged.

If not,

6) Compute ρ_{mixL}, ρ_{mixU}.

7) Go to Step (1).

This procedure is illustrated in Figure 13.

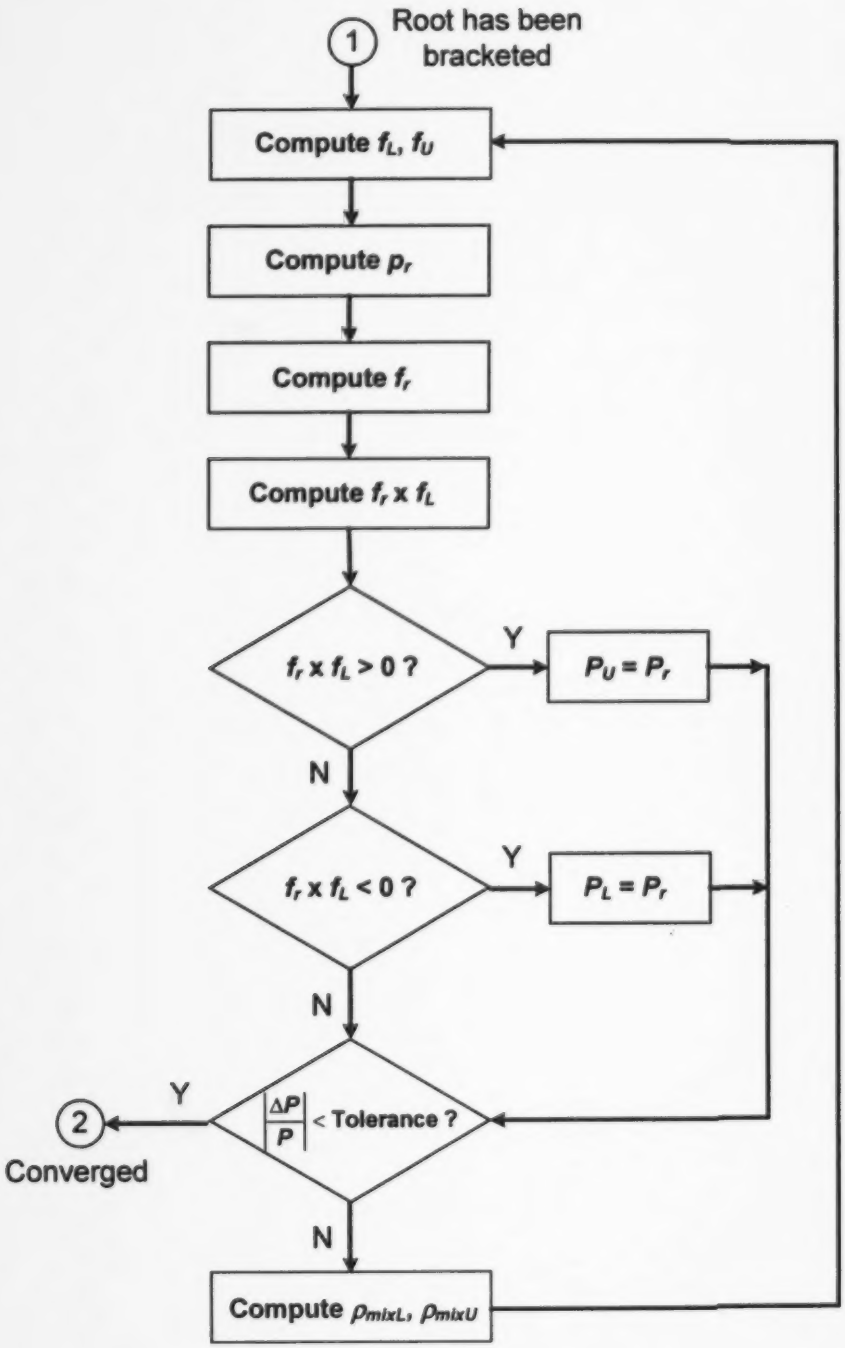


Figure 13: Multiphase Solver Procedure

APPENDIX C MODIFICATIONS TO THE LS-DYNA PENDULUM MODEL

An LS-DYNA finite element model of the pendulum was created as part of a previous contract, and is illustrated in Figure 14. The model was updated in order to properly model the pin running through the clevis holes, as well as the clevis and surrounding plates themselves. The pin and plates are shown in Figure 15.

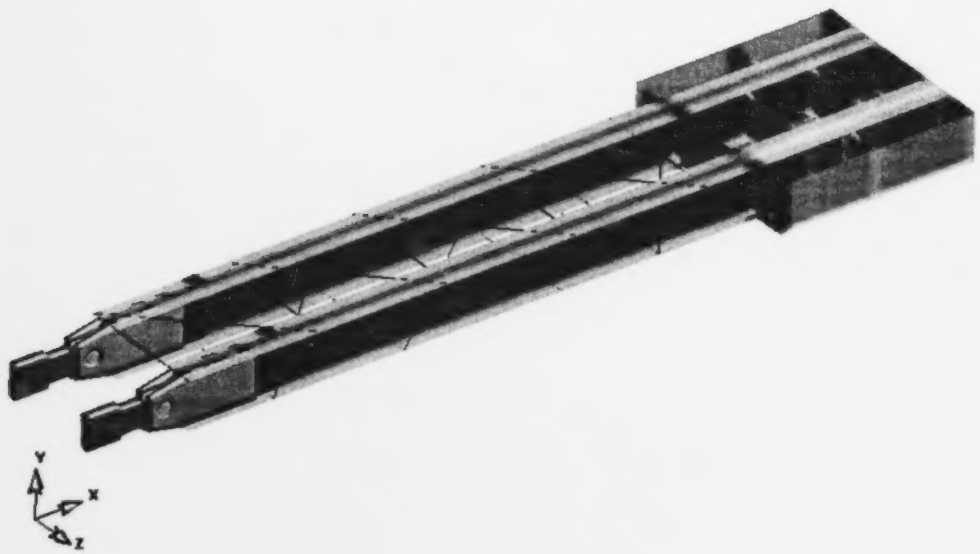


Figure 14: LS-DYNA Pendulum Model

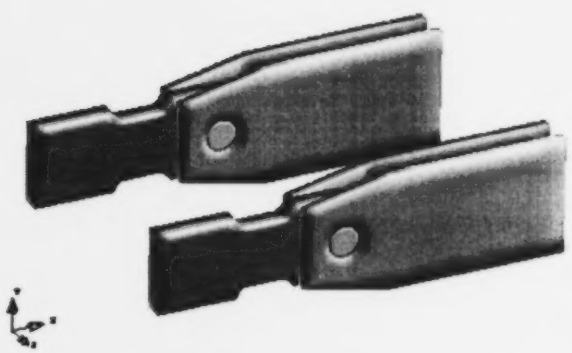


Figure 15: Clevis and Pins

C.1 PIN MODIFICATIONS

The pin from the original model is shown in Figure 16. There are several problems with this component:

1) There is not a node at the centre of the pin although this is the point about which the pendulum rotates.
2) The pin is composed of three pieces rather than one solid piece as in the actual pendulum. These three pieces are connected by a bar element.

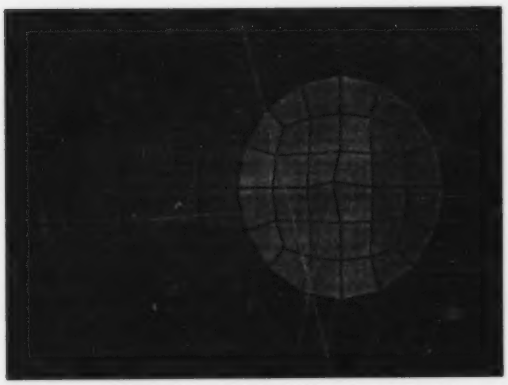


Figure 16: Pin from Original Model

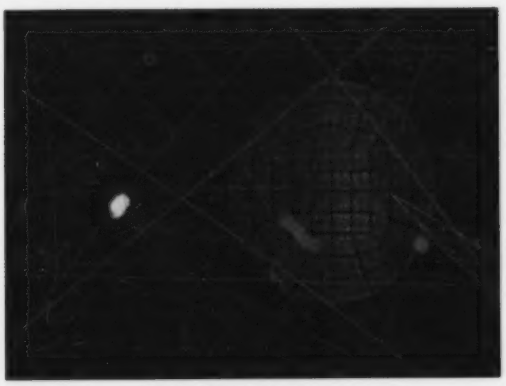


Figure 17: Modified Pin Component

The modified pin element is shown in Figure 17. Modifications made to the original model are:

1) The three individual pin sections and the bar element are replaced by a single solid pin. The diameter of this pin is slightly smaller than in the original model to prevent initial interference with the clevis.
2) The pin was re-meshed such that there is a node located at the pin centre.
3) The contact between the pin and clevis was modified. Two contact groups were defined: a) the contact between the clevis holes and the pin and b) the contact between adjacent clevis surfaces.

C.2 PLATE MODIFICATIONS

The plates from the original model which surround the pendulum pivot point are shown in Figure 18. There is only one element in the thickness, and as a consequence localized bending and torsion effects through the plate thickness could not be adequately captured. These plates were re-meshed with at least three elements through the thickness, as shown in Figure 19.

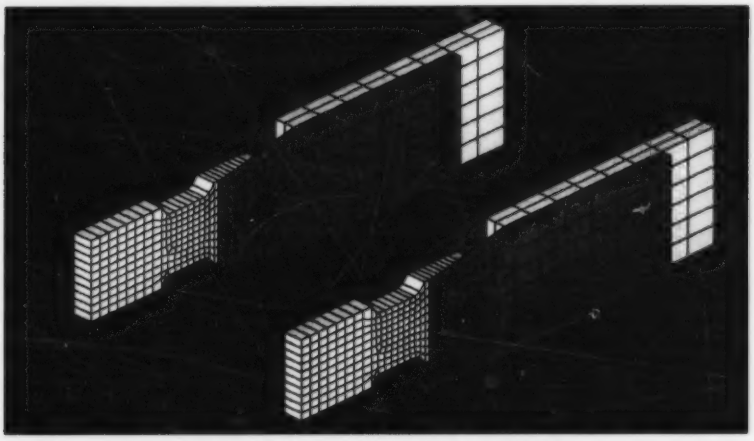


Figure 18: Plates from Original Model

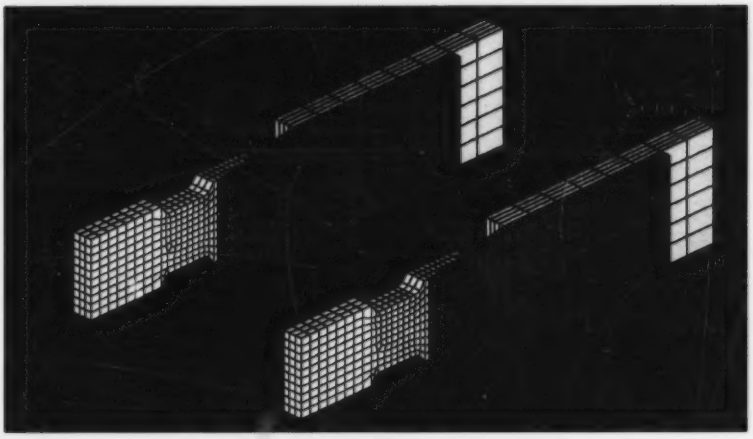


Figure 19: Modified Plate Components

C.3 Other Modifications

While performing the aforementioned changes, a small mistake was also discovered. The gravity loading in the original model was too high by 1 mm/s^2. This has been corrected.

APPENDIX D FSI TUTORIAL

This tutorial will outline how to setup a fluid-structure interaction problem using IFSAS and LS-DYNA. The problem being investigated is shown in Figure 20. An L-shape structure is subjected to a 2 kg TNT blast. It is fixed to the x-y plane (representing the ground). The L-shape is composed of aluminum plates of 2.5 cm thickness.

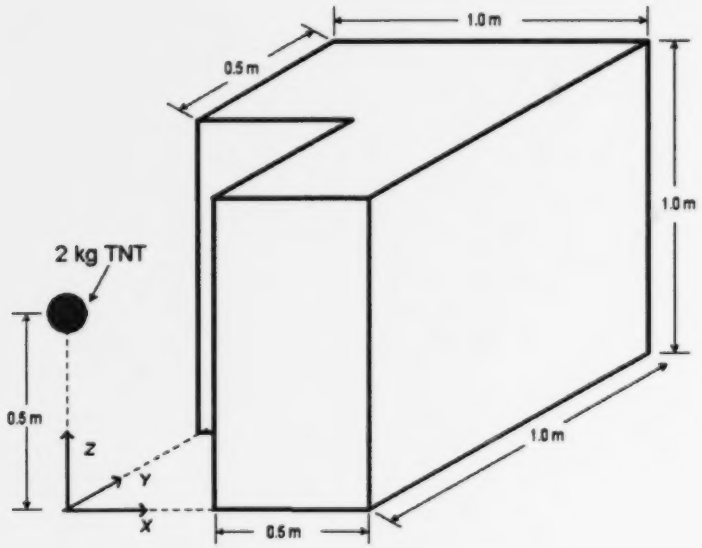


Figure 20: Shape Dimensions

In the coming sections, the L-shape will be created using Martec's Mesh Generator [5] program (see Appendix F for manual). Details on the LS-DYNA input file will be presented, followed by step-by-step instructions on the creation of the IFSAS model.

While this tutorial does contain all of the required information to perform this simulation, there are some terms used within that the user may not be familiar with. More information on the tools and methods used in this tutorial can be found in [6-8], all located in the IFSAS help menu.

D.1 CREATING THE THREE-DIMENSIONAL SHAPE

In this section of the tutorial, a neutral geometry file (NGF) of the L-shape being investigated will be created using the Mesh Generator. The meshed L-shape is shown in Figure 21, and will be formed of thin plates. Note its offset from the IFSAS origin. The shape was placed in this configuration so that an extended grid could be used on the domain boundaries closest to the charge, as will become evident when the IFSAS model is created.

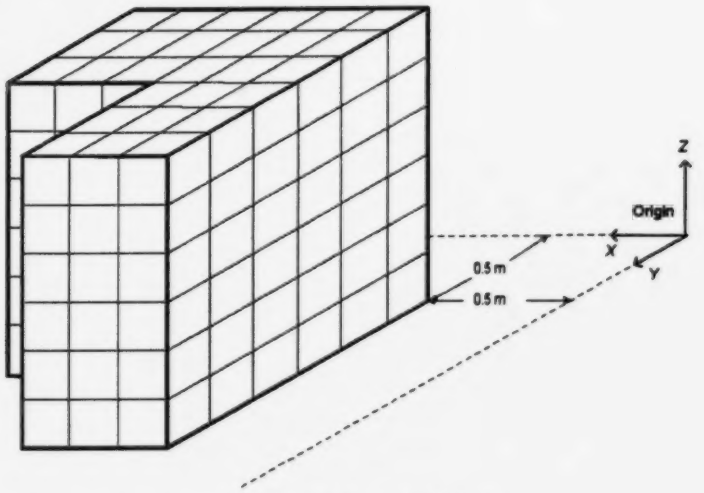


Figure 21: Meshed Shape

Create a new directory (for example, C:\Program Files\Martec\Mesh Generator\lshape)to hold the mesh files. Figure 22 and Figure 23 show the MGN file (Mesh Generator input file) used to recreate this shape. This file can be created in any text editor, and should be saved as *lshape.mgn* in the newly created *lshape* directory. In Card 2.1, 16 grid points are defined indicating the corners of the 14 patches comprising the shape. Figure 24 shows the location of these points on the L-shape.

Patches are then defined using these grid points to create surfaces (Card 2.3). The material type number on Side 1 of the patch is 32002, and is 32001 on Side 2 of the patch. These sides are defined in Figure 25, and are dependant on the order of the grid points defining the patch. The patch material numbers are defined such that the interior of the shape is assigned material number 32001, while the exterior has material number 32002. The material number 32001 was specified on the interior in order to cut down on the IFSAS computation time. IFSAS considers shapes with this material number to be solid shapes, and will therefore not perform computations in the L-shape's inner cells.

The patches are meshed in Card 3.0. The *n_r* and *n_s* fields indicate the number of elements in the r and s directions, respectively. These directions are shown in Figure 25. Note that the directions of *r* and *s* for a patch are dependent upon the order in which the grid points were specified when the patch was created. The *n_t* field is not required since the shape is formed of patches as opposed to hyperpatches.

Finally, in Card 4.0 all patches are joined to form the finished shape.

```
!       Card 1.0:  Problem control variables
!
!       n_grids  n_lines n_patchesn_hypers n_joins  n_meshes   n_start   e_start
        16       0       14       0         1         14         1         1
!
!       Card 1.1: Problem Control Variables 2
!
!       ex_mode   nm_grids  n_timefn
        0         0         0
!
!       Cards 2 -  Geometric entity information
!
!       2.1  Grids
!
!       ent_type      unused
!       grid_no  x1    x2    x3
!       (two lines repeated for each entity defined)
        1         0       0       0
        1         1.0     1.5     0.0
        1         0       0       0
        2         0.5     1.5     0.0
        1         0       0       0
        3         0.5     1.0     0.0
        1         0       0       0
        4         0.5     0.5     0.0
        1         0       0       0
        5         1.0     0.5     0.0
        1         0       0       0
        6         1.5     0.5     0.0
        1         0       0       0
        7         1.5     1.0     0.0
        1         0       0       0
        8         1.0     1.0     0.0
        1         0       0       0
        9         1.0     1.5     1.0
        1         0       0       0
        10        0.5     1.5     1.0
        1         0       0       0
        11        0.5     1.0     1.0
        1         0       0       0
        12        0.5     0.5     1.0
        1         0       0       0
        13        1.0     0.5     1.0
        1         0       0       0
        14        1.5     0.5     1.0
        1         0       0       0
        15        1.5     1.0     1.0
        1         0       0       0
        16        1.0     1.0     1.0
!
!
!       Card 2.3  Patch Enties
!
!       ent_type mat_type
!       ent_no   n1      n2      n3      n4      n5      n6      n7      n8
!       (two lines repeated for each entity defined)
        4         1       32002   32001
        1         1       2       10      9
        4         1       32002   32001
        2         2       3       11      10
        4         1       32002   32001
        3         3       4       12      11
        4         1       32002   32001
        4         4       5       13      12
        4         1       32002   32001
        5         5       6       14      13
        4         1       32002   32001
        6         6       7       15      14
        4         1       32002   32001
        7         7       8       16      15
```

Figure 22: Part 1 of lshape.mgn file

```
        4       1       32002   32001
        8       8       1       9       16
        4       1       32002   32001
        9       9       10      11      16
        4       1       32002   32001
        10      16      11      12      13
        4       1       32002   32001
        11      15      16      13      14
        4       1       32002   32001
        12      1       2       3       8
        4       1       32002   32001
        13      8       3       4       5
        4       1       32002   32001
        14      7       8       5       6
!
!
!       Cards 3.0  Meshing information
!
!       ent_no  n_r     n_s     n_t     a[0]    a[1]    a[2]    bc[0]   bc[1]
        1       3       6       1       1.0     1.0     1.0     0       0
        2       3       6       1       1.0     1.0     1.0     0       0
        3       3       6       1       1.0     1.0     1.0     0       0
        4       3       6       1       1.0     1.0     1.0     0       0
        5       3       6       1       1.0     1.0     1.0     0       0
        6       3       6       1       1.0     1.0     1.0     0       0
        7       3       6       1       1.0     1.0     1.0     0       0
        8       3       6       1       1.0     1.0     1.0     0       0
        9       3       3       1       1.0     1.0     1.0     0       0
        10      3       3       1       1.0     1.0     1.0     0       0
        11      3       3       1       1.0     1.0     1.0     0       0
        12      3       3       1       1.0     1.0     1.0     0       0
        13      3       3       1       1.0     1.0     1.0     0       0
        14      3       3       1       1.0     1.0     1.0     0       0
!
!       Cards 4.0  Joining information
!
!       n_join
        14
!       join_1
        1       2       3       4       5       6       7       8
        9       10      11      12      13      14
```

Figure 23: Part 2 of *lshape.mgn* file

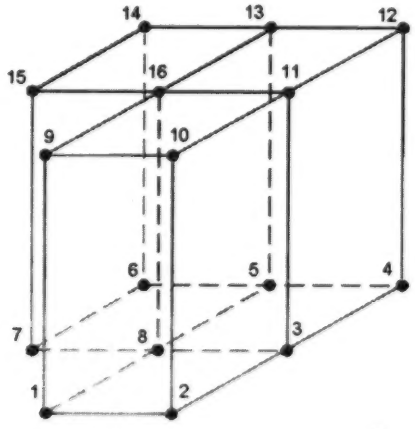


Figure 24: Grid Point Locations

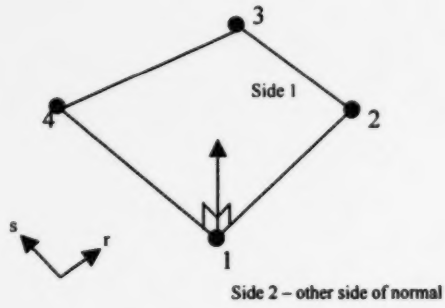


Figure 25: Material Sides for Patches

D.2 EXECUTING THE MESH GENERATOR

The Mesh Generator is executed from the command line using switches. The NGF file created will be called *lshape.ngf*, and the LS-DYNA input file information will be saved as *lshape_LSDYNA.txt*. To perform the conversion, enter the following line in the directory containing the input file. Note that the default paths to the Mesh Generator executable and *lshape* directory are shown here. If IFSAS was installed in another directory, or if the *lshape.mgn* file was saved elsewhere, these paths must be modified accordingly.

```
C:\Program Files\Martec\Mesh Generator\lshape>
            "C:\Program Files\Martec\IFSAS 2002\icfd\mesh.exe" -i lshape.mgn
                  -n lshape.ngf -d lshape_LSDYNA.txt
```

Note: The *-i* switch allows the path and name of the MGN file to be entered (Default value: *mesh_in.dat*). The *-n* switch allows the path and name of the created NGF file to be specified (Default value: *mesh_ngf.dat*). The *-d* switch allows the path and name of the created LS-DYNA input file information to be specified (Default value: *mesh_dyna.dat*). For more details on the Mesh Generator program, see Appendix F.

D.3 CREATING THE LS-DYNA INPUT FILE

This section of the tutorial will briefly outline the creation of the LS-DYNA input file.

When the L-shape MGN file was meshed using the mesh generator, two important files were created: an NGF and an LS-DYNA data file. The LS-DYNA data file, saved as *lshape_lsdyna.txt*, contains data for the following fields of the LS-DYNA input file:

- *ELEMENT_SHELL*
- *DEFINE_CURVE*
- *NODE*

- *DATABASE_HISTORY_NODE*
- *LOAD_SEGMENT*

Add the following keywords and appropriate fields to this file and rename it *lshape.dyn*.

- *CONTROL_TERMINATION*
- *CONTROL_OUTPUT*
- *DATABASE_BINARY_D3PLOT*
- *DATABASE_HISTORY_SHELL*
- *DATABASE_EXTENT_BINARY*
- *MAT_ELASTIC*
- *SECTION_SHELL*
- *PART*

See the completed input file shown at the end of this tutorial for information on specific field settings. Note that the material is aluminum, and all nodes on the lower surface of the shape (z-coord = 0.0) are anchored.

D.4 CREATING THE IFSAS MODEL

In this portion of the tutorial problem the IFSAS model will be created. This 3D model will make use of the IFSAS TNT profiles in order to cut down on the computation time. The L-shape NGF file just created will also be incorporated.

Create a New Run

To begin, a new IFSAS run must be created. Open the IFSAS II GUI and select *File* >> *New* from the menu bar. Enter a run name (Lshape). Select the *3 dimensional* option and press *OK*. The model details must now be specified.

1) Select the *Single Component* model and press *Next* >> to continue.
2) Select the *Ideal Gas* equation of state.
3) Select the *Godunov* solver.
4) Select the *None* in the combustion model window. Press *Finish*.
5) On the model summary page, confirm that the model has been entered correctly and press *OK*.

When a new run is created, a new directory will appear in the IFSAS database (by default located in C:\Program Files\Martec\IFSAS 2002\db). This is the run directory which will contain all the input and output files for the run.

The IFSAS main window will now appear as shown in Figure 26. Icon definitions appear in Figure 27. Note that an icon's definition appears when the cursor is held over an icon for a few seconds.

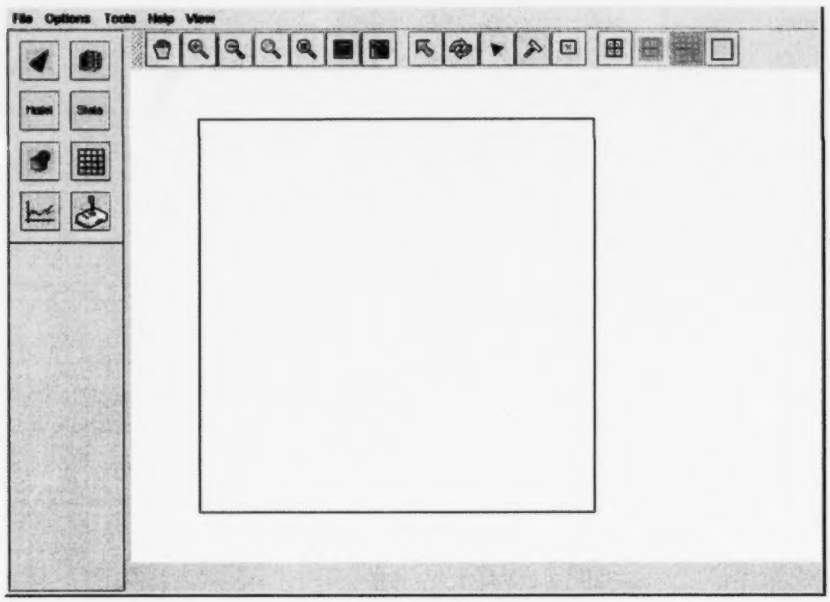


Figure 26: IFSAS GUI

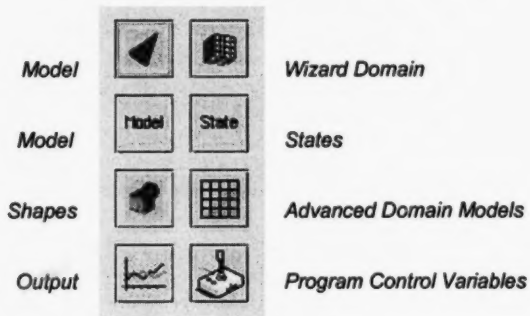


Figure 27: IFSAS GUI Icons

There are eight first-level buttons. Pressing a first-level icon makes available applicable second-level icons. The model wizard has already been specified. Now we will go through the rest of the buttons to define the problem.

NOTE: The first-level buttons are pictured next to the main headings, with the second-level icons pictured next to the appropriate text. Also, icon names referred to in the text appear when the mouse pointer is held over the icon for a couple of seconds.

Define the Domain Specifications

The domain specifications, including grid size and time criteria, must now be defined. Select the *Domain Specifications* icon, and then the *Grid Entry* icon.

1) The domain is 2.0 m x 2.0 m x 1.5 m, with square cells with side length 16.67 cm. This domain will fully encompass the L-shape and the charge. Enter 12 cells in the x direction, 12 cells in the y direction, and 9 cells in the z direction. Then enter the x, y, and z direction domain lengths (2.0 m, 2.0 m, and 1.5 m, respectively).
2) The *Grid Definition* tab is used to modify the x and y geometry. In this example keep the default values (planar).
3) On the *Extended Grid* tab, change the *X-End*, *Y-End* and *Z-End* fields to 5 to define an extended grid around the charge position to be defined.

Now the model time specifications must be entered. Select the *Timing Parameters* icon.

1) On the *Timing Parameters* tab keep the start time of 0 ms and enter an end time of 5.0 ms.
2) On the *Floating Points* tab keep the default courant value of 0.5.

Define the Model Specifications

The definition of the model specifications is accomplished by pressing the *Model* icon and then selecting the *Model Parameters* icon. Confirm that the specific heat ratio is 1.4 and the molecular weight is 28.97 (air).

Define the Material States

Select the *State* icon and then the *Add Fluid State* icon. There are two states required in this model, the ambient state and the explosive state. Only the ambient state needs to be identified here - the explosive state specification will follow. Press the *New* button. Call this state "ambient" and assign the *fluid* state type to it. Enter a pressure of 1 atm and a temperature of 300 K. Press *OK* and close the *Add Fluid State* window.

The explosive state is defined by pressing the *Edit Explosive State* icon. Pick *TNT* from the pull-down menu, and select a charge mass of 2 kg.

It is important to be aware that when performing 3D calculations using the explosive profiles, the mass entered in this window determines the energy applied to the explosive shape volume *within the domain*, not the entire shape. For example, if only half of the explosive shape is located within the domain, then the entire charge mass is applied to the half of the total charge volume located within the domain. This may or may not be desirable, depending on the problem being simulated. In this case the entire charge shape will be located within the domain, therefore the total charge mass is entered in this window. See [8] for more information on using profiles.

The ambient state must also be specified. Press the *Select Ambient State* button (layers), and switch the ambient state to *Ambient*. This will set all areas of the calculation domain not occupied by a shape to this state.

Define the Shapes

Now that the domain state has been set to the ambient conditions, a shape corresponding to the exploded charge must be defined. Press the *Shapes* icon and then the *Shapes* sub-icon. Select the *New* button and call the new shape "Explode", setting the shape to *ellipsoid*. In the shape window set the shape state to *Explosive* and enter the parameters shown in Figure 28.

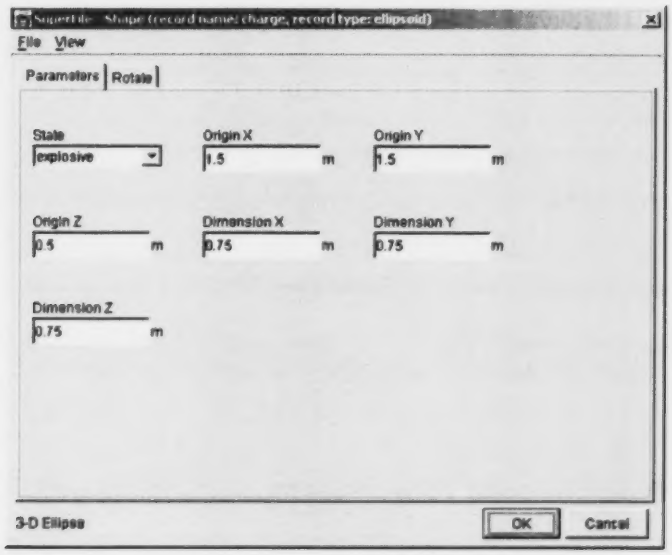


Figure 28: Shape Definition

The L-shape NGF file must now be added. Press *New* and choose the *NGF* type. Pick the NGF (*lshape.ngf*) from the directory containing the L-shape mesh generator files (for example, C:\Program Files\Martec\Mesh Generator\lshape). This NGF will then be saved into the IFSAS run directory. Assign the *solid* state to this shape and press *OK*.

The domain should now appear as shown in Figure 29. The tabs on the top of the graphics window can be used to rotate the axes, change the background colour, and zoom in and out, among other things.

Note the position of the explosive charge. The charge was given a large explosive radius in order to cut down on the calculation time.

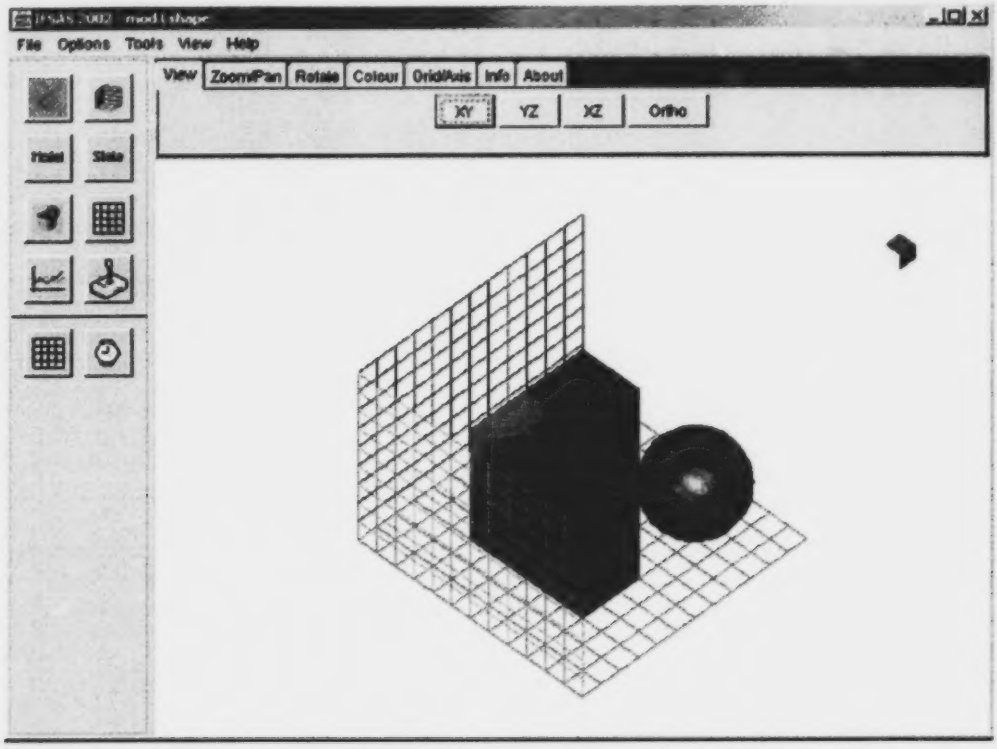


Figure 29: Domain with Shape

Define the Advanced Model Parameters

The edge boundary conditions are set through the *Advanced Domain Models* icon. Select the *Edge Boundary Conditions* icon. Change all the *High* boundaries to *flow through*. The vertical boundaries furthest from the charge will remain solid, as well as the ground.

The LS-DYNA coupling information is specified by pressing the *FEA Configuration* icon. Activate the coupled calculation using the drop-down menu. In the *LS-DYNA Input File* field, enter the name of the LS-DYNA input file ("lshape.dyn"). Finally, highlight the shape corresponding to the target shape from the *Target* list, in this case, *lshape.ngf*. Remember to copy the LS-DYNA input file into the run directory (by default C:\Program Files\Martec\IFSAS 2002\db\lshape).

Define the Output Parameters

The model output parameters must also be defined. Under the *Output Controls* icon, set the time dump flag to *2D regular grid*, and the dump interval to 0.25 ms.

Save the Run

Once all the above information has been entered into the GUI, select *File>>Save* from the menu bar. The run must be saved prior to editing the configuration and input files, and cannot be re-saved once these files have been changed. When the run is saved, changes to the configuration file will be lost.

Edit the IFSAS Input File

A flag must be set manually in the input file to specify that the calculation is one-way coupled. Select the *Tools>>Edit CFD Input* option from the menu bar. If a new window does not appear, check that the file editor is properly specified in the *Utilities* tab of the *Options>>Preferences* menu option. A portion of the input file is shown in Figure 30. In Card 1.7, change the LS-DYNA field from 1 to 3. Save the changes using the text editor's *Save* command and close the window.

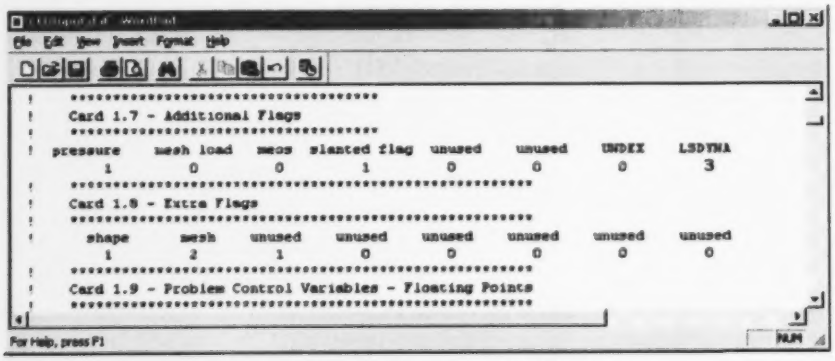

File Edit View Insert Format Help
! ************************************
! Card 1.7 - Additional Flags
! ************************************
! pressure mesh load meos slanted flag unused unused UNDEX LSDYNA
1 0 0 1 0 0 0 3
! **
! Card 1.8 - Extra Flags
! **
! shape mesh unused unused unused unused unused unused
1 2 1 0 0 0 0 0
! **
! Card 1.9 - Problem Control Variables - Floating Points
! **
For Help, press F1
NUM

Figure 30: IFSAS Input File Modifications

Edit the Configuration File / Calculate the Run

The remainder of the tutorial is different depending on the operating system on which LS-DYNA is installed.

CONFIGURATION 1: LS-DYNA on Windows

Select the *Tools>>Edit LSDYNA.cfg* option from the IFSAS menu bar. The LS-DYNA configuration file should appear as shown in Figure 31, but most likely with different paths.

```
C:\LSDYNA\program\ls960-nsmp.exe
C:\Program Files\Martec\IFSAS 2002\db\Lshape
lshape.dyn
0
0
0
1
1
```

Figure 31: Unedited *LSDYNA.cfg* File

The first line of the configuration file states the path and name of the LS-DYNA executable as specified in the *IFSAS.ini* file. The .ini file is located in the *..\IFSAS 2002\res* directory. The second line is the run directory which contains the input files and where output files will be stored. The third line is the name of the LS-DYNA input file. The following three lines are the node, element and load offsets, and are set to zero in this case. The seventh line indicates the number of targets, and the final lines list the IFSAS II shape numbers corresponding to the targets (the first shape in the IFSAS shapes list corresponds to shape number 0).

Important Note: LSDYNA does not accept paths which contain spaces. Therefore, the configuration file must be edited so that this is reflected in the paths. Using the file editor, the configuration file shown in Figure 32 becomes:

```
C:\LSDYNA\program\ls960-nsmp.exe
C:\Progra~1\Martec\IFSAS2~1\db\Lshape
lshape.dyn
0
0
0
1
1
```

Figure 32: Edited *LSDYNA.cfg* File

Note that all paths are expressed in 8.3 format. When using this format, directories are specified using the template *XXXXXX~N* where *XXXXXX* are the first six digits (spaces do not count as digits) of the directory name. The last digit of the format, *N*, is usually equal to 1. However, if there are multiple directories with the same first six digits then N is incremented to indicate the different directories:

C:\IFSAS 2002\ becomes C:\IFSAS2~1
C:\IFSAS 2003\ becomes C:\IFSAS2~2

The windows command line can serve as a check to ensure that the directory names have been converted into 8.3 format correctly. Try changing directories using directory names in 8.3 format as opposed to typing the entire directory name or using wildcards.

Once the changes to the configuration file have been made, save the file using the file editor *Save* command, and close the window.

Now that the configuration file has been saved, the run can be calculated. Select the *File>>Calculate* option. **Note:** DO NOT save the run prior to calculating it. This will undo the changes made to the configuration file and the input file. Currently, each time the run is changed and saved, the configuration file must be updated prior to executing the run.

CONFIGURATION 2: LS-DYNA on Linux

Part I of this section will describe how to set up and perform IFSAS/LS-DYNA coupled FSI calculations on a Linux machine. This only needs to be performed the first time a simulation is set up. For subsequent simulations, skip to Part II.

Part I

On the machine running LS-DYNA, begin by creating a new directory which will contain all required program files for all IFSAS/LS-DYNA coupled runs (for example, */home/user1/IFSAS_FSI*). Directories for all FSI simulations will be placed within this folder.

In addition to any input files, IFSAS requires three components to run: the IFSAS executable, the *Tables* directory, and the *Explosive* directory. In Windows these are all located in *C:\Program Files\Martec\IFSAS 2002\iCFD*. The *Tables* and *Explosive* directories are the same for both Linux and Windows OS. Copy them from the *iCFD* directory into the directory on the Linux machine (*/home/user1/IFSAS_FSI*). The *iCFD* directory also contains an executable compiled for Windows. Martec will supply an executable compiled for Linux as required. Copy the Linux-compiled executable into the *IFSAS_FSI* folder.

Part 2

Create a subdirectory for the simulation within the *IFSAS_FSI* directory (for example, */home/user1/IFSAS_FSI/Lshape*). Avoid directory names containing spaces as LS-DYNA may not accept paths with spaces. Copy the following files from the IFSAS run directory (*C:\Program Files\Martec\IFSAS 2002\db\Lshape*) into the new Linux *Lshape* directory:

- *CFDInput.dat* (the IFSAS input file)
- *lsdyna.cfg* (the FSI configuration file)
- *lshape.dyn* (the LS-DYNA input file)
- *lshape.ngf* (the L-shape geometry file)

The LS-DYNA configuration file, *lsdyna.cfg*, must now be modified to reflect the Linux paths. This file can be opened in any text editor. The first line of the configuration file states the path and name of the LS-DYNA executable as specified in the *IFSAS.ini* file. The .ini file is located in the *..\IFSAS 2002\res* directory. The second line is the run directory which contains the input files and where output files will be stored. The third line is the name of the LS-DYNA input file. The following three lines are the node, element and load offsets, and are set to zero in this case. The seventh line indicates the number of targets, and the final lines list the IFSAS II shape numbers corresponding to the targets (the first shape in the IFSAS shapes list corresponds to shape number 0).

The modified configuration file should appear similar to Figure 33.

```
/home/user1/lsdyna/program/ls960-nsmp.exe
/home/user1/IFSAS_FSI/Lshape
lshape.dyn
0
0
0
1
1
```

Figure 33: Edited *LSDYNA.cfg* File

Copy the executable located in the */home/user1/IFSAS_FSI* folder into the *Lshape* directory. It is good practice to keep a copy of the executable used to run a simulation in the same directory as the input and resulting output files. Not only is this the default location for the executable (meaning the path to the executable does not need to be specified), but it ensures that all old results can be reproduced.

The simulation can now be run. At the prompt, change into the *Lshape* directory and enter the following command:

```
>./ifsas_II -t ../Tables -e ../Explosive -i CFDInput.dat
```

The IFSAS run will begin as in any non-FSI run, however during the IFSAS run, calls to LS-DYNA will be indicated by "Call to LS-DYNA" and "Return Call From LS-DYNA" lines, with LS-DYNA dump information in between.

FSI TUTORIAL ADDENDUM: THE LS-DYNA INPUT FILE

Following are the contents of the LS-DYNA input file, *lshape.dyn*:

```
*KEYWORD
*TITLE
IFSAS - LSDYNA FSI Demo
*CONTROL_TERMINATION
$         f         i         f         f         f
$    endtim    endcyc     dtmin    endneg    endmas
 0.000E-03
*CONTROL_OUTPUT
$         i         i         i         i         f         i         i
$     npopt    neecho    nrefup    iaccop     opifs    ipnint    ikedit
         0         0         0         0                 2      1000
$
$     database section
$
*DATABASE_BINARY_D3PLOT
$         f         i
$  dt/cycl      lcdt
 2.500E-04
*DATABASE_HISTORY_NODE
       1
       2
       3
       4
       5
```

BREAK

```
     195
     196
     197
     198
     199
     200
*DATABASE_HISTORY_SHELL
$         i         i         i         i         i         i         i         i
$       id1       id2       id3       id4       id5       id6       id7       id8
         1
*DATABASE_EXTENT_BINARY
$         i         i         i         i         i         i         i         i
$     neiph     neips    maxint    strflg    sigflg    epsflg    rltflg    engflg
         0                   3                                       2         2
$         i         i         i
$    cmpflg    ieverp    beamip                                     2
$
$     material section
$
*MAT_ELASTIC
$         i         f         f         f         f         f         f
$       mid       rho         e        pr      sigy      etan      beta
         1  2.66E+03  70.3E+09      0.33   322E+06   340E+06  0.00E+00
*SECTION_SHELL
$         i         i         f         f         f         f         i
$       sid    elform      shrf       nip     propt   qr/irid     icomp
         1        16       0.0         5                 0.0
$         f         f         f         f         f
$        t1        t2        t3        t4      nloc
 2.500E-02 2.500E-02 2.500E-02 2.500E-02 0.000E+000
$
$     part section
$
*PART
Kinematic/Isotropic elastic-plastic
```

```
$       i         i         i         i         i         i         i
$     pid       sid       mid     eosid      hgid     igrav    adpopt
        1         1         1         0         0         0         0
$
$     mesh section
$
*NODE
       1   1.000000e+000   1.500000e+000   0.000000e+000       7       0
       2   8.333333e-001   1.500000e+000   0.000000e+000       7       0
       3   6.666667e-001   1.500000e+000   0.000000e+000       7       0
       4   5.000000e-001   1.500000e+000   0.000000e+000       7       0
       5   1.000000e+000   1.500000e+000   1.666667e-001       0       0
```

BREAK

```
     195   8.333333e-001   6.666667e-001   0.000000e+000       7       0
     196   6.666667e-001   6.666667e-001   0.000000e+000       7       0
     197   1.333333e+000   8.333333e-001   0.000000e+000       7       0
     198   1.166667e+000   8.333333e-001   0.000000e+000       7       0
     199   1.333333e+000   6.666667e-001   0.000000e+000       7       0
     200   1.166667e+000   6.666667e-001   0.000000e+000       7       0
*ELEMENT_SHELL
       1       1       1       2       6       5
       2       1       2       3       7       6
       3       1       3       4       8       7
       4       1       5       6      10       9
       5       1       6       7      11      10
```

BREAK

```
     194       1     197     198     200     199
     195       1     198     191     194     200
     196       1     113     199      93      94
     197       1     199     200      92      93
     198       1     200     194      73      92
$
$     define section
$
*DEFINE_CURVE
       1
$       i         i         f         f         f         f
$     lcid      sidr      scla      sclo      offa      offo
$                 f                   f
$           abscissa           ordinate
          0.000E+000         1.000E+000
          1.000E+000         1.000E+000
*DEFINE_CURVE
       2
$       i         i         f         f         f         f
$     lcid      sidr      scla      sclo      offa      offo
$                 f                   f
$           abscissa           ordinate
          0.000E+000         1.000E+000
          1.000E+000         1.000E+000
*DEFINE_CURVE
       3
$       i         i         f         f         f         f
$     lcid      sidr      scla      sclo      offa      offo
$                 f                   f
$           abscissa           ordinate
          0.000E+000         1.000E+000
          1.000E+000         1.000E+000
```

BREAK

```
*DEFINE_CURVE
     394
$       i         i         f         f         f         f
$     lcid      sidr      scla      sclo      offa      offo
$                 f                   f
$           abscissa           ordinate
          0.000E+000         1.000E+000
```

```
          1.000E+000          1.000E+000
*DEFINE_CURVE
       395
$        i         i         f         f         f         f
$     lcid      sidr      scla      sclo      offa      offo
$                  f                   f
$             abscissa            ordinate
          0.000E+000          1.000E+000
          1.000E+000          1.000E+000
*DEFINE_CURVE
       396
$        i         i         f         f         f         f
$     lcid      sidr      scla      sclo      offa      offo
$                  f                   f
$             abscissa            ordinate
          0.000E+000          1.000E+000
          1.000E+000          1.000E+000
*LOAD_SEGMENT
$        i         f         f         i         i         i         i
$     lcid        sf        at        n1        n2        n3        n4
         1  1.0E+00  0.0E+00         1         2         6         6
         2  1.0E+00  0.0E+00         6         5         1         1
         3  1.0E+00  0.0E+00         2         3         7         7
         4  1.0E+00  0.0E+00         7         6         2         2
         5  1.0E+00  0.0E+00         3         4         8         8
```

BREAK

```
       392  1.0E+00  0.0E+00        93        94       113       113
       393  1.0E+00  0.0E+00       199       200        92        92
       394  1.0E+00  0.0E+00        92        93       199       199
       395  1.0E+00  0.0E+00       200       194        73        73
       396  1.0E+00  0.0E+00        73        92       200       200
*END
```

APPENDIX E IFSAS TUTORIAL

This tutorial gives step-by-step instructions on the creation of the landmine model used to generate the reported results. In this example the 40 cm standoff, 5 cm overburden, 6% moisture case will be detailed.

In this simulation, the IFSAS II Thrust Calculator will be used to determine the impulse loading on the plate. As a consequence, the target plate must be modelled using a neutral geometry file (NGF), the creation of which is outlined below. Also, since the Thrust Calculator currently only measures forces in the x-direction in axisymmetric problems, the entire domain will be rotated 90° CCW for the simulation, as shown in Figure 34.

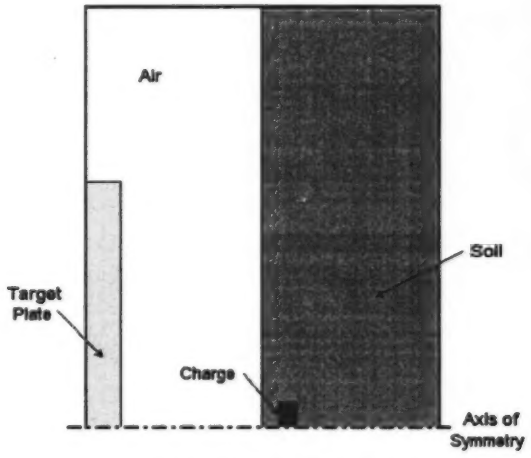


Figure 34: Rotated Domain

E.1 NGF File Creation

The target plate must be represented by an NGF shape in order for the Thrust Calculator to determine the forces acting upon it. NGF files can be created using Martec's Mesh Generator program (see Appendix F). Use of this program is detailed in [5,6]. Since in this case the shape is very simple (four nodes connected by four lines), the NGF file will be created manually. Using a text editor, create a file with the lines of information shown in Figure 35.

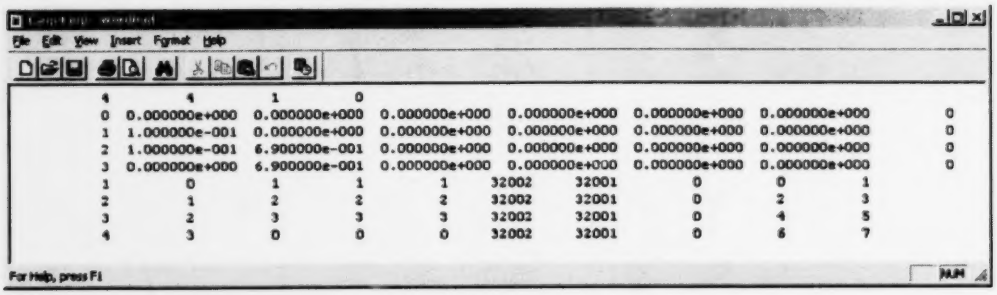


Figure 35: *Target.ngf* file

Save the file as *target.ngf*. In the coming section, this file will be imported into the CFD calculation.

E.2 IFSAS MODEL GENERATION

The present section will outline the creation of the landmine simulation using the IFSAS II multiple material model.

Create a New Run

Open the IFSAS II GUI and select *File* >> *New* from the menu bar. Enter a run name (Landmine in 6% moisture soil), and select the *2 dimensional* option.

1) Select the *Multiple Material* model and press *Finish* to continue.
2) On the model summary page press *OK*.

The IFSAS main window will now appear as shown in Figure 36. Icon definitions appear in Figure 37. Note that an icon's definition appears when the cursor is held over an icon for a few seconds.

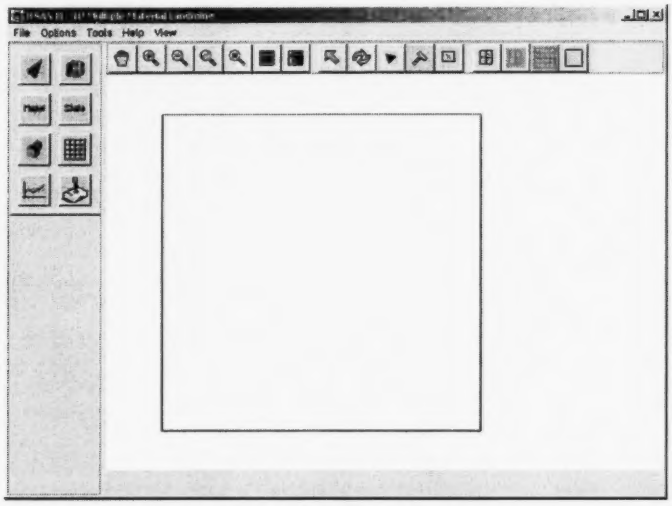


Figure 36: IFSAS GUI

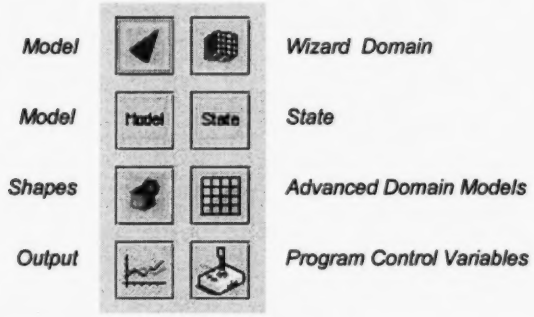


Figure 37: IFSAS GUI Icons

There are eight first-level buttons. Pressing a first-level icon makes available applicable second-level icons. The model wizard has already been specified. Now we will go through the rest of the buttons to define the problem.

NOTE: The first-level buttons are pictured next to the main headings, with the second-level icons pictured next to the appropriate text. Also, icon names referred to in the text appear when the mouse pointer is held over the icon for a couple of seconds.

Define the Domain Specifications

The domain specifications, including grid size and time criteria, must now be defined. Select the *Domain* icon, and then the *Grid Entry* icon pictured on the left.

1) On the *Grid Information* tab enter 150 cells in the x-direction and an x-distance of 1.5 m and 120 cells in the y-direction and a y-distance of 1.2 m.

2) The *Grid Definition* tab is used to modify the x and y geometries. Since this problem is axisymmetric, change the change the y geometry to *cylindrical*.

3) The *Extended Grid* tab allows cells outside of the main calculation domain to be specified. In this case keep the default values.

Now the model time specifications must be entered. Select the *Timing Parameters* icon (shown on the left).

1) On the *Timing Parameters* tab keep the start time of 0 ms and enter an end time of 10.0 ms.

2) On the *Floating Points* tab keep the default Courant value of 0.5.

Define the Model Specifications

For this simple model, the model specification includes only the model definition. Select the *Model* icon and then the *Model Parameters* icon shown on the left. On the *Gas* tab, change the *Initial Density* value to that calculated using Equation (21), and on the *Water* and *Soil* tabs change the *Exponent m* value to 2.0. Confirm all other values shown in Table 4.

Table 4: Material Properties

Tab Name	Parameter	Value
Gas	Gas Constant	287.1848
	Global Specific Heat Ratio	1.4
	Molecular Weight (g/mol)	28.97
	Initial Density (kg/m^3)	1000
	Initial Pressure (atm)	1.0
	Initial Enthalpy (J/kg)	104899.4
	Ideal Gas Initial Pressure (atm)	1.0
	Ideal Gas Initial Density (kg/m^3)	0.906
	Ideal Gas Gamma	1.4
Water	Tait Exponent m	2.0
	Tait Exponent n	7.14
	Tait Exponent B	3.0498825E8
	Water Initial Pressure (atm)	1.0
	Water Initial Density (kg/m^3)	1000
Soil	Soil Bulk Modulus	3.37745098E10
	Soil Initial Density (kg/m^3)	2650
	Soil Exponent m	2.0

Define the Material States

Select the *State* icon. First the material states must be entered. Select the *Add Fluid State* icon (pictured to the left), and press the *New* button to add a new state. Call this state "Ambient Air" and enter the state information shown in Table 5. Repeat to create two additional states, "Soil" and "Mine". The soil state incorporates the calculated soil volume fractions in Table 3, while the mine state uses the pressure calculated in Equation (20).

Table 5: State Definitions

Parameter	Ambient Air	Soil	Mine
Velocity 1 (m/s)	0	0	0
Velocity 2 (m/s)	0	0	0
Velocity 3 (m/s)	0	0	0
Pressure (atm)	1	1	39959.5
Temperature (K)	300	300	300
Volume Fraction (Gas)	1	0.437	1
Volume Fraction (Water)	0	0.078	0
Volume Fraction (Solid)	0	0.485	0
Tracker Front Status	Pre-front	Pre-front	Pre-front

The ambient state must also be defined. Press the *Select Ambient State* button shown on the left, and switch the ambient state to *Ambient Air*. This will set all areas of the calculation domain not occupied by a shape to this state.

Define the Shapes

Now that the domain state has been set to the ambient conditions, a shape corresponding to the high pressure state must be defined. Press the *Shape* icon, and then the *Shapes* sub-icon. Select the *New* button and call the new shape "Soil", and set the shape to *rectangle*. In the shape window, enter the parameters shown below in Table 6. Repeat for the *Target* and *Mine* shapes. Note that to import the NGF file, simply select the file from the directory structure. The NGF file is then copied into the run directory.

Table 6: Shape Definitions

Parameter	Soil	Target	Mine
Type	Rectangle	NGF	Rectangle
State	Soil	Solid	Mine
X Origin (m)	1.0	N/A	0.575
Y Origin (m)	0.6		0.03
X Dimension (m)	0.99999		0.04999
Y Dimension (m)	1.19999		0.05999

The domain should now appear as shown in Figure 38. Note the NGF shape indicated by the solid line.

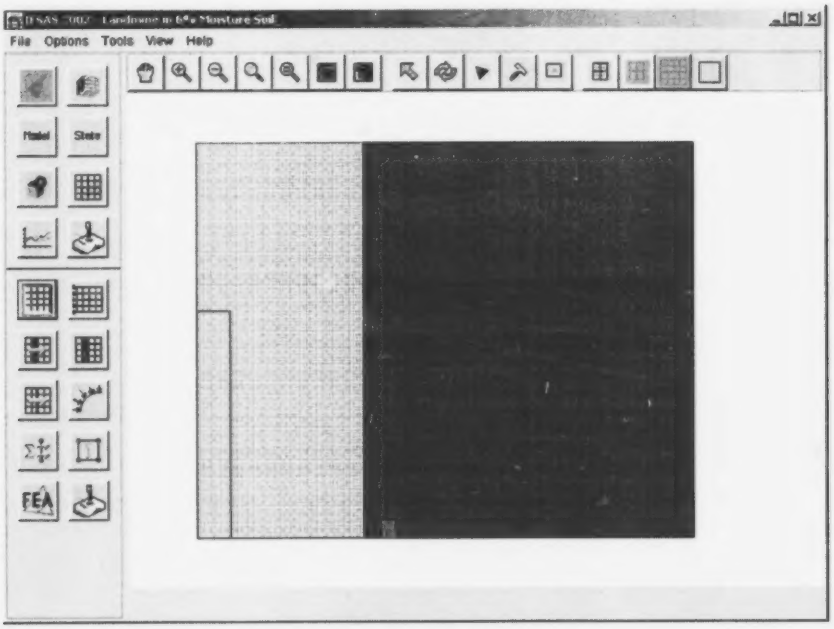


Figure 38: Domain with Shapes

Define the Advanced Domain Model Parameters

The edge boundary conditions are set through the *Advanced Domain Models* icon. Press the *Edge Boundary Conditions* icon. Change the *X Low Boundary, X High Boundary* and the *Y High Boundary* to *flow through* to allow the solution to pass through the ends and upper side of the domain. Press *OK*.

The Thrust Calculator must be activated in order that the force on the target plate be measured. Press the *Thrust Calculation* icon, and then the *New* button. Create a new thrust calculation criteria called "target". In the Thrust Calculator window, select the *target.ngf* shape from the list since the forces on this shape are of interest. Select the *search all cells* option from the drop-down menu. In nearly all Thrust Calculator applications, the *search all cells* is the option required.

Also under the *Advanced Domain Models* icon, press the control icon depicted on the left, and change the tracker flag to *Inactive*.

Define the Output Parameters

The model output parameters must be defined. Under the *Output* icon, set the time dump flag to *2D Regular Grid*, and the dump interval to 0.5 ms. On the *Mesh Loads* tab, enter a dump interval of 1E-5 s. This is the interval at which Thrust Calculator data is written to the *force_sum.dat* file.

Save the Run

Save the model by selecting *File >> Save Run* from the menu bar.

Change Solver Parameters

Currently the solver parameters cannot be modified from the GUI. Therefore the input file must be modified. Note: Changes to the input file are undone when the model is saved from the GUI (*File>>Save* option). Therefore, ensure that the model is saved prior to editing the input file.

The input file is edited by selecting *Tools >> Edit CFD Input* from the menu bar. A new text editor window should appear. If the input file does not appear, check that the text editor path is correctly specified on the *Utilities* tab of the *Options >> Preferences* menu item.

Make the changes/additions to Card 11 highlighted in Figure 39 below. These changes to the solver parameters are required to prevent solution instabilities. Then save the input file using the file editor's *Save* command.

Calculate the Run

The run is calculated by selecting *File >> Calculate Run*. Do not use the GUI to save the run prior to running it - this will undo the changes to the input file.

Viewing the Results

The main CFD results can be viewed using the *File >> Output* option which will launch TecPlot and open the *udmtec.dat* file. The Thrust Calculator output is stored in the *force_sum.dat* file located in the same directory (the run directory), and must be opened manually in TecPlot.

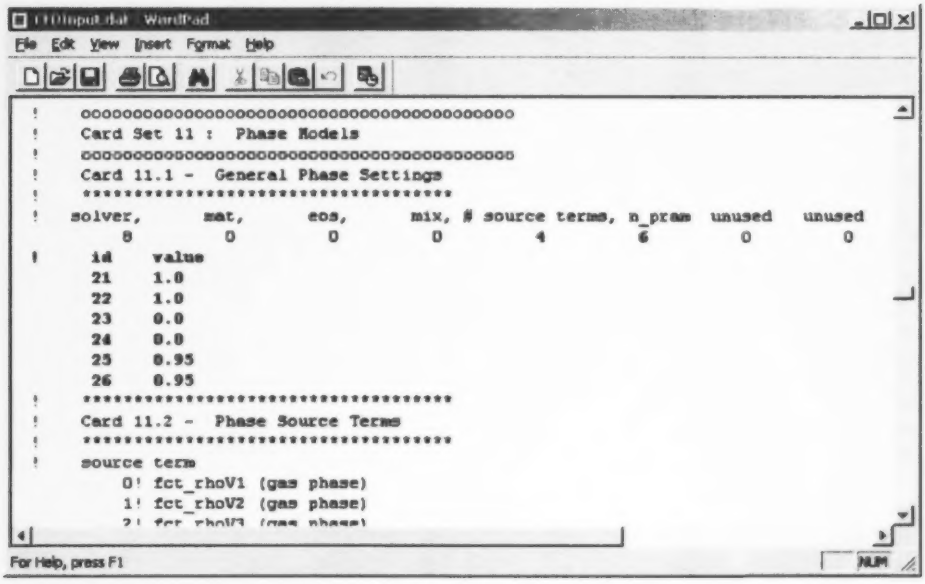


Figure 39: Changes to Input File

APPENDIX F MESH GENERATOR INPUT MANUAL

The following document is the manual for the Martec's Mesh Generator program.

Program freda

A Mesh Generation Program For Finite Element Meshes and CFD Grids

By

Rick Link
Combustion Dynamics Ltd.

Revision 1.3
June 21, 2001

1.0 Introduction

The program freda is a mesh generation program for finite element and CFD grids. Current capabilities include:

1) 2 node beams, 4 node shells, and 8 node hexahedrons.

2) Linear and curved geometries.

3) Assignment of displacement boundary conditions and part numbers.

4) Joining (node deletion) of any element type.

Nodes and elements for the computational model are generated using a reference geometry called a **solid entity**. Examples of solid entities are shown in Figure 1.1. The nodes are first mapped onto the solid entity space. Element connectivities are then generated (see Figure 1.2). Currently supported solid entities include:

1) Linear and cubic lines.

2) Bilinear and bicubic patches.

3) Trilinear and tricubic hyperpatches.

Complex shapes are created by joining surface nodes together, as shown in Figure 1.3. This ensures variable continuity across entity boundaries.

There is currently I/O to five different files:

1) Input file – has all the input data described in the manual.

2) Output file – echo checks the input data.

3) TECPLOT file – writes a TECPLOT plotting file. Currently supports all element types.

4) LSDYNA file – writes nodal and element data in LSDYNA keyword format.

5) NGF file – writes nodal and element data in Neutral Geometry Format (NGF).

The mesh generator code can be run from a command line interface. The user has the options of specifying I/O filenames, using default filenames, or being prompted for filenames at run time. For example typing:

```
mesh -i  input_filename -t tecplot_filename
```

specifies the input and tecplot filenames to be used. Typing

```
mesh -i input_filename
```

tells the program to use input_filename as the input file and to use the default output filename. Typing:

```
mesh
```

(or running the program from a compiler) runs the mesh generator code, prompting the user for all of the filenames. Typing

```
mesh -D
```

tells the program to use all of the default filenames. Typing:

```
mesh -L
```

displays a list of default filenames. Typing

```
mesh -T time
```

allows input of a calculation time, used in prescribed motion problems with the udm code. Finally, typing:

```
mesh ?          OR          mesh -?
```

displays a help screen (which includes all of the available argument options and a brief description of how to use command line keys). If an invalid argument is entered an error message appears which instructs the user to type "mesh ?" for help.

Switches and Default Filenames used by the Mesh Generator

Switches	Description	Default Filename
-I	Specifies input filename	mesh_in.dat
-o	Specifies echo check filename	mesh.out
-t	Specifies TECPLOT filename	mesh_tec.dat
-d	Specifies LSDYNA filename	mesh_dyna.dat
-n	**Specifies NGF filename** *	mesh_ngf.dat
-D	Uses default files	
-L	Lists default Filenames	
-T	Specifies time for prescribed motion	
? OR -?	Displays Help Screen	

* only supports bilinear patches and lines

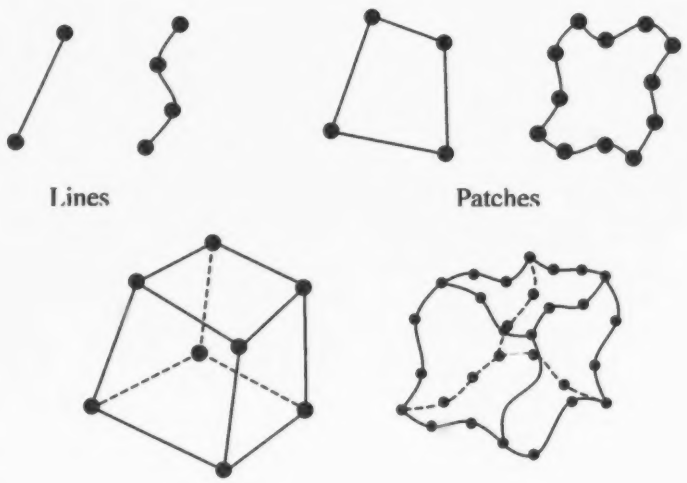
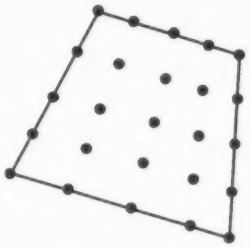
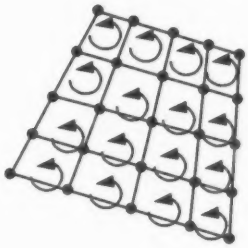


Figure 1.1 Geometric entites

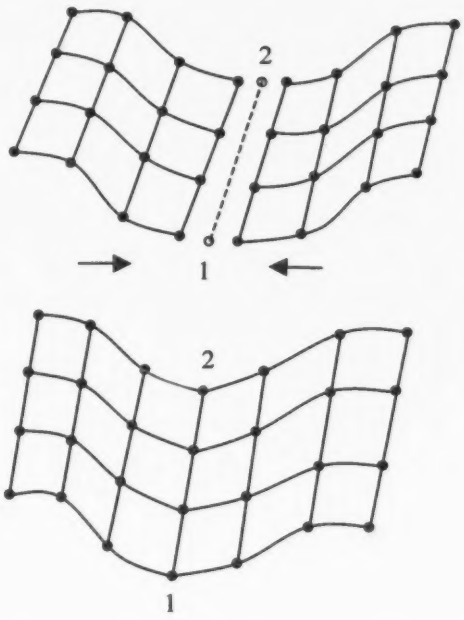


Step 1: Nodal generation



Step 2: Element generation
Nodal connectivities

Figure 1.2 Model generation



Joining done by specifying coincident corner grids

Figure 1.3 Joining nodes

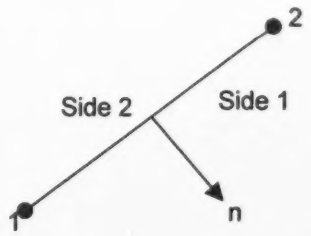
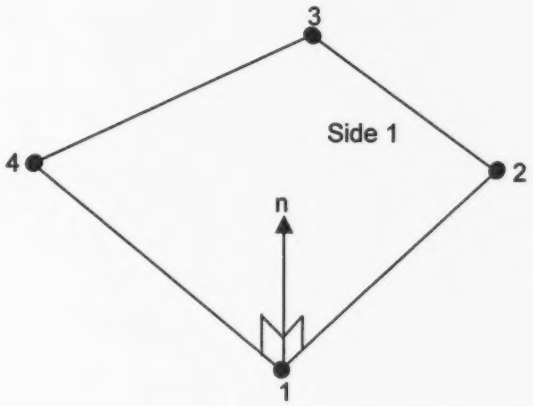


Figure 1.4a Material Sides for Lines



Side 2 – other side of normal

Figure 1.4b Material Sides for Patches

Figures 1.4 Side 1 and Side 2 for Material Numbers

2.0 Input Cards

Card 1.0: Problem Control Variables (8I10)

Description	Format
Number of grids	I10
Number of lines	I10
Number of patches	I10
Number of hyperpatches	I10
Number of joins	I10
Number of meshed entities	I10
Start node	I10
Start element	I10

Card 1.1: Problem Control Variables 2 (3I10)

Description	Format
Execution mode = 0 Normal generation = 1 Moving mesh coupled with CFD code	I10
Number of movable grids	I10
Number of time functions	I10

Cards 2.0: Geometric Entity Information (2I10)

Enter this card for each geometric entity.

Description	Format
Entity type = 1 Grid = 2 Linear line (3) = 3 Cubic line (5) = 4 Bilinear patch (4) = 5 Bicubic patch (12) = 6 Trilinear hyperpatch (8) = 7 Tricubic hyperpatch (32)	I10
Part number	I10
Material Number Side 1	I10
Material Number Side 2	I10

NOTE: The material information is used primarily for CFD application input. If not used for a CFD application enter –1 for Material Numbers.

Material Numbers are only valid for Lines and Patches, but entries are required for all Entity Types.
(See Figure 1.4 for order of Material Sides)

Notes:

1) Solid entities are defined as entity types 2-7.
2) Number in parentheses indicate the number of grids to describe the entity.

Cards 2.1: Grid Entity (I10, 3D15)

Description	Format
Solid entity number	I10
x1 - coordinate	I10
x2 - coordinate	I10
x3 - coordinate	I10

Cards 2.2: Line Entity (6I10)

Description	Format
Solid entity number	I10
Grid 1	I10
Grid 2	I10
Grid 3	I10
Grid 4	I10
Grid 5	I10

Notes:

1) If the line type is a linear line (entity type = 2), only enter grids 1-3. Grids 3 for a linear line, and 5 for a cubic line are orientation nodes, commonly used in finite element beam definitions (see Figure 2.1)

Cards 2.3 Patch Entity (9I10)

Description	Format
Solid entity number	I10
Grid 1	I10
Grid 2	I10
Grid 3	I10
Grid 4	I10
Grid 5	I10
Grid 6	I10
Grid 7	I10
Grid 8	I10

Notes:

1) For bilinear patches, enter 4 grid numbers. For bicubic patches, enter 12 grid numbers, with grid 9 starting on the next line.

Cards 2.4 Hyperpatch Entity (9I10)

Description	Format
Solid entity number	I10
Grid 1	I10
Grid 2	I10
Grid 3	I10
Grid 4	I10
Grid 5	I10
Grid 6	I10
Grid 7	I10
Grid 8	I10

Notes:

1) For bilinear patches, enter 4 grid numbers. For bicubic patches, enter 32 grid numbers, with grid 9 starting on the next line. Lines are read in formats of 9I10.

Card 3.0 Meshing Information (4I10, 3D15, 2I10)

Description	Format
Entity number	I10
Number of r-elements	I10
Number of s-elements	I10
Number of t-elements	I10
Expand ratio 1	D15
Expand ratio 2	D15
Expand ratio 3	D15
Translational boundary condition	I10
Rotational boundary condition	I10

Notes:

1) The directions r, s, and t are the natural (curved) coordinate system of the entity, as shown in Figure 2.2.

2) The expand ratios generate nodes according to a geometric progression. For example, if the ratio was 2, the nodes would be generated as follows:

Node 3: 2x node 2
Node 4: 4x node 2
Node 5: 8x node 2....etc.

Cards 4.0 Joining Information

Description	Format
Number of solid entities to join	I10

Cards 4.1 Joining Information 2 (8I10)

Description	Format
Solid entity number 1	I10
Solid entity number 2	I10
Solid entity number 3	I10
Solid entity number 4	I10
Solid entity number 5	I10
Solid entity number 6	I10
Solid entity number 7	I10
Solid entity number 8	I10

Notes:

1) An unlimited number of entities may be joined. The entity numbers are input on lines of format 8I10.

Cards 5.0 Movable Grid Information

Description	Format
Grid number	I10
Time function number	I10
Position type = 0 Displacement = 1 Grid coordinates	I10

Cards 6.0 Time Function Information

Repeat for number of time functions in Card 1.1.

Card 6.1 Time Function Data (2I10)

Description	Format
Time function number	I10
Number of points	I10

Card 6.2 Time Function Data 2 (4D15)

Description	Format
Time	D15
Value 1	D15
Value 2	D15
Value 3	D15

Card 7.0: Additional Problem Control Variables (8I10)

Description	Format
Number of load segments	I10
Unused	I10
Unused	I10
Unused	I10
Unused	I10
Unused	I10
Unused	I10
Unused	I10

Cards 8.0 Load Calculation Information

Card 8.1 Load Calculation Data (2I10)

Repeat for number of load segments.The load segment group number is made up of a number of entities and the sum of the forces on these entities will be the force on the load segment.

Description	Format
Load segment group number	I10
Number of entities in section	I10

Card 8.2 Load Calculation Data 2 (XI10)

Repeat for number of entities that make up the load segment group number.

Description	Format
Entity number (from card 3.0)	I10

Node 3 defines the plane from which the beam axes can be oriented.

Figure 2.1 Beam orientation nodes

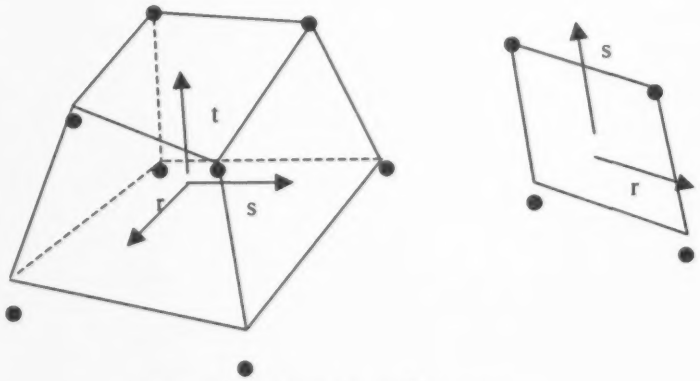


Figure 2.2 Natural coordinate system

3.0 Example: Three Patch Mesh

This input file is an example of 3 patches with a mesh resolution of 5x5 elements each. The patches are bicubic, and are joined at the common boundaries. Figure 3.1 is a plot of the final meshed shape.

```
!
!       test18.dat
!       Curved patch - multiple patches - join test
!       5x5 mesh for each patch
!

!
!       Card 1.0:  Problem control variables
!
!  n_grids   n_lines n_patches  n_hypers   n_joins  n_meshes   n_start   e_start
!
        25         0         3         0         1         3         1         1
!
!       Card 1.1:  Problem control variables
!
!  ex_mode  nm_grids n_timefun
!
         0         0         0
!
!       Cards 2:  Geometric entity information
!
!       2.1  Grids
!
! ent_type
!
         1         0        -1        -1
!
!  grid_no             x1             x2             x3
!
         1            1.0            0.0            0.0
         1              0             -1             -1
         2      0.9659258      0.2588190            0.0
         1              0             -1             -1
         3      0.8660254            0.5            0.0
         1              0             -1             -1
         4      0.7071067      0.7071067            0.0
         1              0             -1             -1
         5            0.5      0.8660254            0.0
         1              0             -1             -1
         6      0.2588190      0.9659258            0.0
         1              0             -1             -1
         7            0.0            1.0            0.0
         1              0             -1             -1
         8            0.8            0.0            0.0
         1              0             -1             -1
         9      0.6047378      0.6047378            0.0
         1              0             -1             -1
        10            0.0            0.8            0.0
         1              0             -1             -1
        11            0.6            0.0            0.0
         1              0             -1             -1
        12      0.5023689      0.5023689            0.0
         1              0             -1             -1
        13            0.0            0.6            0.0
         1              0             -1             -1
        14            0.4            0.0            0.0
         1              0             -1             -1
        15            0.4      0.1333333            0.0
         1              0             -1             -1
        16            0.4      0.2666667            0.0
         1              0             -1             -1
        17            0.4            0.4            0.0
         1              0             -1             -1
        18      0.2666667            0.4            0.0
         1              0             -1             -1
        19      0.1333333            0.4            0.0
         1              0             -1             -1
        20            0.0            0.4            0.0
         1              0             -1             -1
        21      0.2666667            0.0            0.0
         1              0             -1             -1
        22      0.1333333            0.0            0.0
         1              0             -1             -1
        23            0.0            0.0            0.0
         1              0             -1             -1
        24            0.0      0.1333333            0.0
         1              0             -1             -1
        25            0.0      0.2666667            0.0
!
!       Cards 2.4  Patch entities
!
! ent_type
!
         5         1        -1        -1
!
!   ent_no        n1        n2        n3        n4        n5        n6        n7        n8
!
         1         1         4        17        14         2         3         9        12
        16        15        11         8
         5         1        -1        -1
         2         4         7        20        17         5         6        10        13
        19        18        12         9
         5         1        -1        -1
         3        17        20        23        14        18        19        25        24
```

```
      22        21        15        16
!
!     Cards 3.0  Meshing information
!
!  ent_no       n_r       n_s       n_t          a[0]          a[1]          a[2]     bc[0]     bc[1]
!
       1         5         5         0           1.0           1.0           1.0         0         0
       2         5         5         0           1.0           1.0           1.0         0         0
       3         5         5         0           1.0           1.0           1.0         0         0
!
!     Cards 4.0  Joining information
!
!  n_join
!       3
!
!   ent_1     ent_2     ent_3
!       1         2         3
```

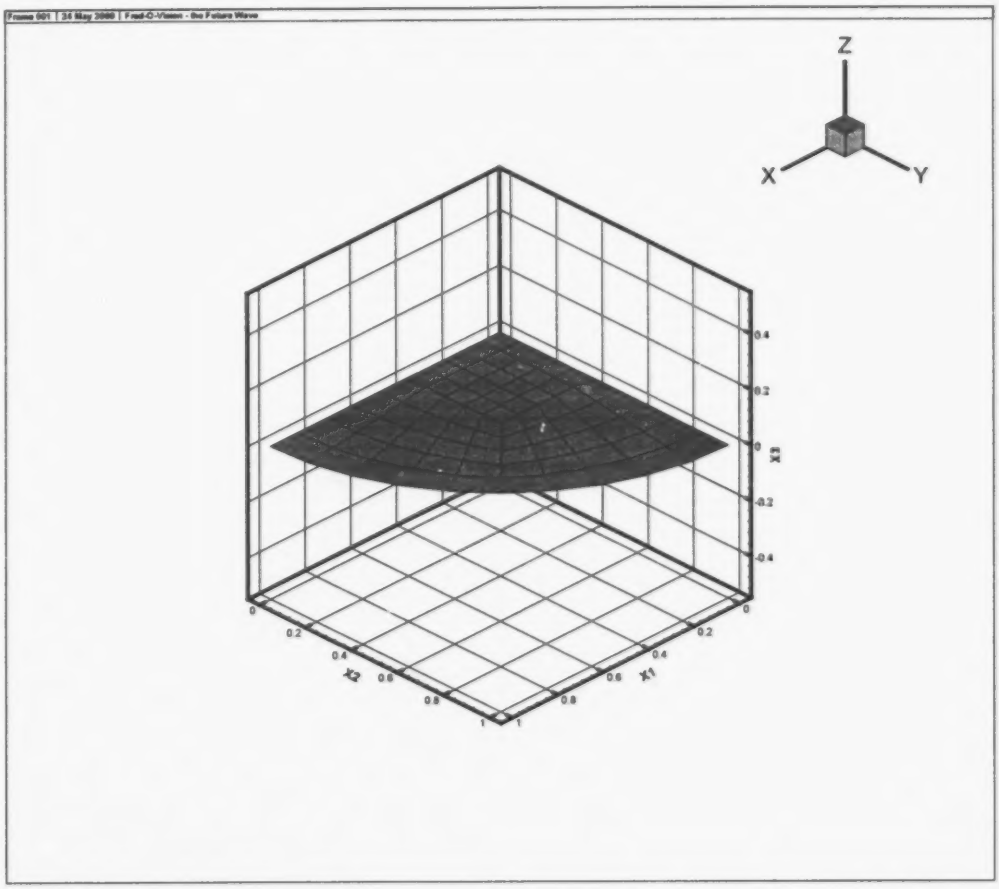


Figure 3.1 Final meshed shape

4.0 Example: Moving Mesh

This input file is an example of a variable geometry nozzle for a pulse detonation engine. It creates a two-dimensional geometry consisting of points connected by lines. A time function is defined so that the code can be run with the –T command line switch. Figure 4.1 shows nozzle geometries at the start time and at 0.2 s and 0.4 s.

```
!
!       test1.dat
!       flap-like line mesh
!
!
!       Card 1:  Problem control variables
!
!  n_grids   n_lines n_patches  n_hypers   n_joins  n_meshes   n_start   e_start
!       16        18         0         0         0        18         1         1
!
!       Card 1.1 PCV's 2
!
! ex_mode   nm_grids  n_timefn
!        1         4         4
!
!       Cards 2:  Geometric entity information
!
!       2.1  Grids
!
! ent_type
!        1         0        -1        -1
!
!  grid_no             x1                x2                x3
!
         1               0.0               0.4               0.0
         1         0     0.0        -1     -1
         2               0.0               0.5               0.0
         1         0     2.0        -1     -1
         3               2.0               0.5               0.0
         1         0     3.0        -1     -1
         4               3.0               0.5               0.0
         1         0     3.0        -1     -1
         5               3.0               0.4               0.0
         1         0     2.0        -1     -1
         6               2.0               0.4               0.0
         1         0     0.0        -1     -1
         7               0.0              -0.5               0.0
         1         0     0.0        -1     -1
         8               0.0              -0.4               0.0
         1         0     2.0        -1     -1
         9               2.0              -0.4               0.0
         1         0     3.0        -1     -1
        10               3.0              -0.4               0.0
         1         0     3.0        -1     -1
        11               3.0              -0.5               0.0
         1         0     2.0        -1     -1
        12               2.0              -0.5               0.0
         1         0     3.0        -1     -1
        13               3.0               0.5               0.0
         1         0     3.0        -1     -1
        14               3.0               0.4               0.0
         1         0     3.0        -1     -1
        15               3.0              -0.4               0.0
         1         0     3.0        -1     -1
        16               3.0              -0.5               0.0
!
!       Cards 2.2  line entities
!
! ent_type  mat_type
!        2         1        -1        -1
!
!   ent_no        n1        n2        n3        n4        n5        n6        n7        n8
!
         1         1         2         1
         2         1        -1        -1
         2         2         3         2
         2         1        -1        -1
         3         3         4         3
         2         1        -1        -1
         4         4         5         4
         2         1        -1        -1
         5         5         6         5
         2         1        -1        -1
         6         6         1         6
         2         1        -1        -1
         7         7         8         7
         2         1        -1        -1
         8         8         9         8
         2         1        -1        -1
         9         9        10         9
         2         1        -1        -1
        10        10        11        10
         2         1        -1        -1
        11        11        12        11
         2         1        -1        -1
        12        12         7        12
         2         1        -1        -1
```

```
        13          3         13          3
         2          1         -1         -1
        14         13         14         13
         2          1         -1         -1
        15         14          6         14
         2          1         -1         -1
        16          9         15          9
         2          1         -1         -1
        17         15         16         15
         2          1         -1         -1
        18         16         12         16
!
!       Cards 3.0  Meshing information
!
!   ent_no       n_r       n_s       n_t          a[0]          a[1]          a[2]    bc[0]    bc[1]
!
         1        20         1         1           1.0           1.0           1.0        0        0
         2        20         1         1           1.0           1.0           1.0        0        0
         3        20         1         1           1.0           1.0           1.0        0        0
         4        20         1         1           1.0           1.0           1.0        0        0
         5        20         1         1           1.0           1.0           1.0        0        0
         6        20         1         1           1.0           1.0           1.0        0        0
         7        20         1         1           1.0           1.0           1.0        0        0
         8        20         1         1           1.0           1.0           1.0        0        0
         9        20         1         1           1.0           1.0           1.0        0        0
        10        20         1         1           1.0           1.0           1.0        0        0
        11        20         1         1           1.0           1.0           1.0        0        0
        12        20         1         1           1.0           1.0           1.0        0        0
        13        20         1         1           1.0           1.0           1.0        0        0
        14        20         1         1           1.0           1.0           1.0        0        0
        15        20         1         1           1.0           1.0           1.0        0        0
        16        20         1         1           1.0           1.0           1.0        0        0
        17        20         1         1           1.0           1.0           1.0        0        0
        18        20         1         1           1.0           1.0           1.0        0        0
!
!       Cards 4.0  Joining information
!

!       Cards 5.0 Movable Grid Information
!  grid_no timefn_no posn_type
         4         1         1
         5         2         1
        10         4         1
        11         3         1

!       Cards 6.0 Time Function Information
!timefn_no no_points
         1         7
!          time          value1          value2          value3
           0.0             3.0             0.5             0.0
           0.1          2.9962          0.4126             0.0
           0.2          2.9848          0.3258             0.0
           0.3          2.9659          0.2404             0.0
           0.4          2.9397          0.1569             0.0
           0.5          2.9063          0.0761             0.0
           0.6          2.8660         -0.0014             0.0
         2         7
           0.0             3.0             0.4             0.0
           0.1          2.9875          0.3132             0.0
           0.2          2.9674          0.2279             0.0
           0.3          2.9400          0.1446             0.0
           0.4          2.9055          0.0640             0.0
           0.5          2.8640         -0.0133             0.0
           0.6          2.8160         -0.0866             0.0
         3         7
           0.0             3.0            -0.5             0.0
           0.1          2.9962         -0.4126             0.0
           0.2          2.9848         -0.3258             0.0
           0.3          2.9659         -0.2404             0.0
           0.4          2.9397         -0.1569             0.0
           0.5          2.9063         -0.0761             0.0
           0.6          2.8660          0.0014             0.0
         4         7
           0.0             3.0            -0.4             0.0
           0.1          2.9875         -0.3132             0.0
           0.2          2.9674         -0.2279             0.0
           0.3          2.9400         -0.1446             0.0
           0.4          2.9055         -0.0640             0.0
           0.5          2.8640          0.0133             0.0
           0.6          2.8160          0.0866             0.0
```

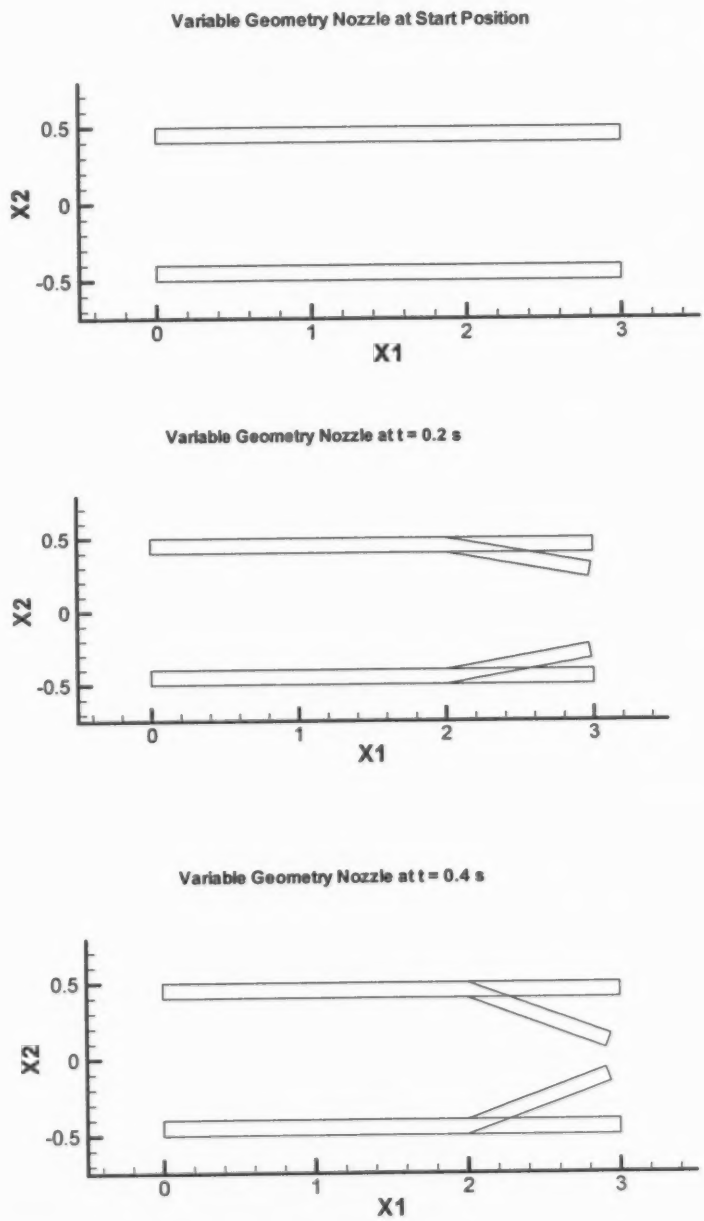


Figure 4.1 Nozzle position at start time and at time = 0.2 s and 0.4 s.

UNCLASSIFIED

SECURITY CLASSIFICATION OF FORM

(highest classification of Title, Abstract, Keywords)

DOCUMENT CONTROL DATA

(Security classification of title, body of abstract and indexing annotation must be entered when the overall document is classified)

1. ORIGINATOR (the name and address of the organization preparing the document. Organizations for who the document was prepared, e.g. Establishment sponsoring a contractor's report, or tasking agency, are entered in Section 8.)

   Martec Limited
   Suite 400 – 1888 Brunswick Street
   Halifax, NS B3J 3J8

2. SECURITY CLASSIFICATION (overall security classification of the document, including special warning terms if applicable)

   UNCLASSIFIED

3. TITLE (the complete document title as indicated on the title page. Its classification should be indicated by the appropriate abbreviation (S, C or U) in parentheses after the title).

   Numerical Characterization of Mine Blast Loading (U)

4. AUTHORS (Last name, first name, middle initial. If military, show rank, e.g. Doe, Maj. John E.)

   Martin, Laura K.; Link, Rick A.

5. DATE OF PUBLICATION (month and year of publication of document)

   August 2003

6a. NO. OF PAGES (total containing information, include Annexes, Appendices, etc) 82

6b. NO. OF REFS (total cited in document) 8

7. DESCRIPTIVE NOTES (the category of the document, e.g. technical report, technical note or memorandum. If appropriate, enter the type of report, e.g. interim, progress, summary, annual or final. Give the inclusive dates when a specific reporting period is covered.)

   Contract Report (Final Report)

8. SPONSORING ACTIVITY (the name of the department project office or laboratory sponsoring the research and development. Include the address.)

   DRDC Suffield/Military Engineering Section, PO Box 4000, Station Main, Medicine Hat, AB T1A 8K6

9a. PROJECT OR GRANT NO. (If appropriate, the applicable research and development project or grant number under which the document was written. Please specify whether project or grant.)

9b. CONTRACT NO. (If appropriate, the applicable number under which the document was written.)

   W7702-01-R883

10a. ORIGINATOR'S DOCUMENT NUMBER (the official document number by which the document is identified by the originating activity. This number must be unique to this document.)

   Martec Report #TR-03-18

10b. OTHER DOCUMENT NOs. (Any other numbers which may be assigned this document either by the originator or by the sponsor.)

   DRDC Suffield CR 2006-016

11. DOCUMENT AVAILABILITY (any limitations on further dissemination of the document, other than those imposed by security classification)

   (x) Unlimited distribution
   ( ) Distribution limited to defence departments and defence contractors; further distribution only as approved
   ( ) Distribution limited to defence departments and Canadian defence contractors; further distribution only as approved
   ( ) Distribution limited to government departments and agencies; further distribution only as approved
   ( ) Distribution limited to defence departments; further distribution only as approved
   ( ) Other (please specify):

12. DOCUMENT ANNOUNCEMENT (any limitation to the bibliographic announcement of this document. This will normally corresponded to the Document Availability (11). However, where further distribution (beyond the audience specified in 11) is possible, a wider announcement audience may be selected).

13. ABSTRACT (a brief and factual summary of the document. It may also appear elsewhere in the body of the document itself. It is highly desirable that the abstract of classified documents be unclassified. Each paragraph of the abstract shall begin with an indication of the security classification of the information in the paragraph (unless the document itself is unclassified) represented as (S), (C) or (U). It is not necessary to include here abstracts in both official languages unless the text is bilingual).

This report presents results from a numerical characterization of mine blast loading performed using experimental data obtained from DRDC Suffield small pendulum apparatus trials.

The validation of the IFSAS multiple material model was performed with data for two landmine trials representing the extremes in soil moisture content. The dependence of the applied impulse on the specific heat ratio and the soil stiffness was investigated.

Results from the landmine model are qualitatively and quantitatively consistent with experimental results. The trend of increasing impulse with increasing moisture content was reproduced. The specific heat ratio was found to have little effect on the calculated impulse. Also, the soil stiffness was decreased from its default value for sand in an attempt to determine the effect of other lower-stiffness soil components such as clay. Results indicate a significant dependence on soil stiffness, and demonstrate the need for more detailed soil information prior to simulating detonations in soils.

14. KEYWORDS, DESCRIPTORS or IDENTIFIERS (technically meaningful terms or short phrases that characterize a document and could be helpful in cataloguing the document. They should be selected so that no security classification is required. Identifies, such as equipment model designation, trade name, military project code name, geographic location may also be included. If possible keywords should be selected from a published thesaurus, e.g. Thesaurus of Engineering and Scientific Terms (TEST) and that thesaurus-identified. If it is not possible to select indexing terms which are Unclassified, the classification of each should be indicated as with the title.)

Landmine, Simulation, Soil Model